MADCAP User's Guide*

The NPARC Alliance

NASA Glenn Research Center
Cleveland, Ohio

USAF Arnold Engineering Development Center
Tullahoma, Tennessee

*This document describes the use of MADCAP Version 2.15. It is essentially the same as the help files distributed with MADCAP, written by Mark Fisher, Todd Michal, and Andrew Cary of Boeing Phantom Works. Please send documentation suggestions/corrections to Charlie Towne at *towne@grc.nasa.gov*. Questions about Wind-US itself or the NPARC Alliance should be sent to *nparc-support@info.arnold.af.mil*, (931) 454-7885

Contents

1 Introduction **1**

2 Graphics Screen **3**
2.1 Name List . . . 3
2.1.1 General Information . . . 3
2.1.2 *.cgd* Files . . . 5
2.1.3 *.csf* Files . . . 8
2.1.4 *.tmp* Files . . . 9
2.2 Screen Rotation and Translation Controls . . . 9
2.3 Main Display Area . . . 11
2.4 Message Area . . . 11
2.5 Dial Box and Spaceball . . . 12
2.6 Mouse Buttons . . . 12
2.6.1 Picking . . . 13
2.6.2 Centering . . . 16
2.6.3 Selecting . . . 16
2.6.4 Toggle Surface Off . . . 16
2.6.5 Toggle Surface On . . . 16
2.6.6 Turn Surface Off . . . 16

3 File Menu **17**
3.1 New . . . 17
3.2 Open . . . 18
3.2.1 IGES Files . . . 19
3.3 Close . . . 21
3.4 Import . . . 21
3.4.1 Import Grid File . . . 21
3.4.2 Import Geometry File . . . 22
3.5 Export . . . 23
3.6 Rename Objects . . . 24
3.7 Delete Objects . . . 25
3.8 Transfer Objects . . . 25
3.9 Conversions . . . 28
3.9.1 Temp to CSF Conversion . . . 28
3.9.2 PLOT3D to CSF Conversion . . . 29
3.9.3 AFLR to CSF Conversion . . . 29
3.9.4 AFLR to CGD Conversion . . . 30
3.10 Save State . . . 30
3.11 Run Script . . . 30
3.12 Set FS,BL,WL Directions . . . 31
3.13 Set File Units . . . 32
3.14 Set File Global Scale . . . 33
3.15 Quit . . . 34

4 View Menu **37**
4.1 Reset . . . 37
4.2 Clear Screen . . . 37
4.3 View Angles . . . 37
4.4 Views . . . 38

4.5 Screen Capture ... 40
4.6 Global Display Mode 40
4.7 New Object Display Mode 41
4.8 Unstruct Edge Extraction Mode 42
4.9 Wire Frame Sequencing 42
4.10 Points on Curves 43
4.11 Highlight Object Origins 44
4.12 Blanking Mode 45
4.13 Color 46
4.14 Light Settings 48
4.15 Sweep Plane Settings 49
4.16 Axis Orientation on Reset ... 51
4.17 Surface File Sort Mode 52
4.18 Axis Mode 52
4.19 Set Output Lines Per Page 53

5 Create Menu **55**
5.1 Create New Zone 55
5.2 Assign Surface to Zone Plane 56
5.3 Create Curve 57
5.3.1 Manual Input 57
5.3.2 Line 59
5.3.3 Circle 60
5.3.4 Ellipse 62
5.3.5 Cubic/Quadratic 63
5.3.6 Centroid 65
5.3.7 Cut Plane 67
5.4 Create Surface 68
5.4.1 Body of Revolution 68
5.4.2 Stack Curves/Surfaces 70
5.4.3 Build Between Curves 71
5.4.4 Centroid 73
5.5 Create Intersection 75
5.6 Create Projection 76
5.7 Create Singularity 78
5.8 Create Copy 79
5.8.1 Unstructured Object Options 80
5.9 Create Extension 81

6 Manipulate Menu **83**
6.1 Break Objects 83
6.2 Join Objects 84
6.3 Insert Object (Structured Grids Only) 86
6.4 Transform Phys. Coords 87
6.5 Transform Comp. Coords (Structured Grids Only) 89
6.6 Rotate Objects 90
6.7 Translate Objects 92
6.8 Scale Objects 93
6.9 Reflect Objects 94
6.10 Move Points 96
6.11 Delete Indices 97

6.12 Add Indices (Structured Grids Only) . . . 98
6.13 Abut Surface Edges (Structured Grids Only) . . . 99
6.14 Abut Loop Edges . . . 100
6.15 Flip/Align Surface Normals (Unstructured Grids Only) . . . 101
6.16 Swap Surface Edges (Unstructured Grids Only) . . . 102
6.17 Merge Points (Unstructured Grids Only) . . . 103
6.18 Edit Unstructured Object (Unstructured Grids Only) . . . 104
6.19 Convert Structured to Unstructured . . . 105

7 Grid Generation Menu **107**
7.1 2D TFI . . . 107
7.2 Regrid Surface or Curve . . . 111
7.2.1 Equal Spacing . . . 112
7.2.2 Hyperbolic Tangent . . . 112
7.2.3 Hyperbolic Sine . . . 113
7.2.4 Hybrid HypTan/HypSin . . . 114
7.2.5 Geometric Progression . . . 115
7.2.6 Match Existing Distribution . . . 115
7.2.7 Maintain Existing Distribution . . . 116
7.2.8 Sine/Cosine Functions . . . 116
7.2.9 Same as Side 1/2 . . . 116
7.2.10 Surface Regrid Examples . . . 116
7.3 Map Surface or Curve onto Another . . . 117
7.4 Unstructured Mesh Quality . . . 118
7.5 2D Surface Elliptic . . . 119
7.6 Surface/Curve Smoothing . . . 122

8 Overlapping Grid Menu **123**
8.1 Set Overlapping File . . . 123
8.2 Set Overlapping Region . . . 124
8.3 Define Cutter . . . 125
8.3.1 Define Surface Cutters . . . 125
8.3.2 Define Index Cutters . . . 127
8.3.3 Define Punch Cutters . . . 129
8.4 Define Boundary Surface . . . 130
8.4.1 Define Fringe Boundary Surface . . . 131
8.4.2 Define Zone Face Boundary Surface . . . 131
8.4.3 Define Boundary Surface by Index . . . 132
8.5 Generate Hole . . . 134
8.6 Generate Boundary Surface . . . 134
8.7 Modify Hole . . . 135
8.8 Modify Boundary Surface . . . 136
8.9 Print Overlapping Information . . . 137
8.10 Update Overlapping Data . . . 138
8.11 Cancel Overlapping Mods . . . 138
8.12 Release Overlapping File . . . 138

9 Boundary Condition Menu **139**
9.1 Boundary Condition Sequence . . . 139
9.2 Identify Points . . . 143
9.3 Auto-couple . . . 144

9.4 Set Tolerances 145
9.5 Set Coupling Mode 146
9.6 Set Coupling Interpolation Mode 147
9.7 Set Boundary Layer Coupling 147
9.8 Set Capture Area 148
9.9 Print BC Information 149
9.10 Update Boundary Conditions 150
9.11 Cancel Boundary Condition Mods 150
9.12 Release Boundary Condition File 150

10 Macro Menu **151**
10.1 Create Outer Boundary 151
10.2 Development Macros 151
10.2.1 Draw CGD Boundaries 152
10.3 Personal Macros 152

Appendix A. MADCAP Keywords **153**

1 Introduction

MADCAP (Modular Aerodynamic Design Computational Analysis Process) is a tool developed in the Applied CFD group at Boeing Phantom Works in St. Louis. When fully implemented, this system will give the Computational Fluid Dynamics (CFD) user access to the full range of tools and processes used to perform CFD analyses under one user interface, from geometry acquisition through grid generation, flow solver setup and execution, and post-processing of results. While the user interface and structure of the system are new, the underlying structured grid work packages initially consists largely of the ZONI3G, GMAN, and CFPOST tools already familiar to users. A significant amount of new technology has been added to support unstructured grids. In the future, other tools may be included in the system as appropriate. MADCAP utilizes a graphical user interface (GUI) built on a *Tcl/Tk* framework, as well as the MDGL graphics library for 3D graphics. MADCAP users may view *.cgd* files, the standard grid file in use by Wind-US, and *.csf* files, the new common file format for surface data previously held in the ZONI3G *.tmp* file. A utility to convert files from the *.tmp* format to the *.csf* format is also available within the system. MADCAP can work directly on analytic geometry imported from an IGES file. Several geometry file formats can be imported into MADCAP through the CADfix program. Several viewing capabilities are available in MADCAP which are not available in the current set of CFD tools. A full description of these capabilities is available in Section 4.

The user drives MADCAP via a menu bar displayed across the top of the screen, as shown below.

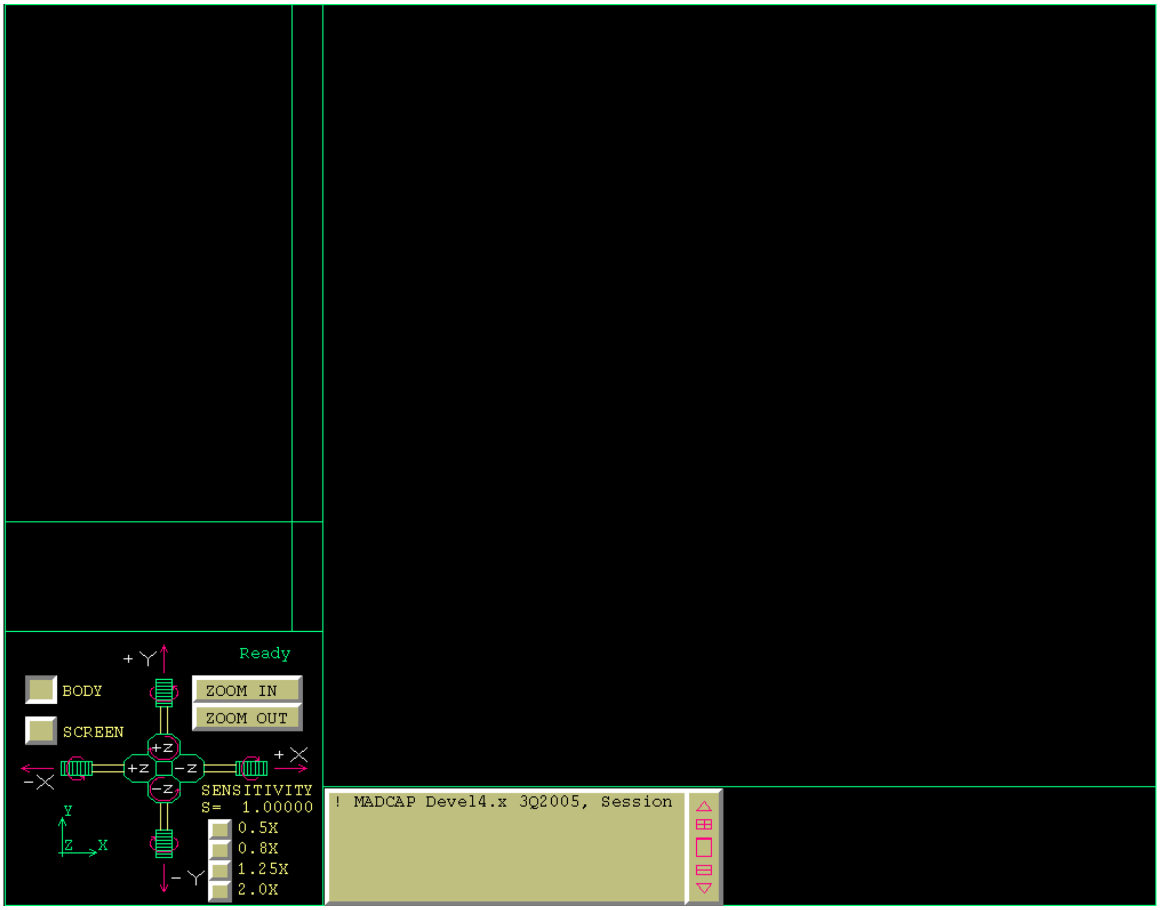
These menus provide access to all of the various toolsets available in MADCAP. Most of these toolsets access independent software libraries utilized by the MADCAP system.

Drop-down menus for the menu choices on the main menu bar are displayed when the user selects a menu item using the left mouse button. These menus may consist of additional cascaded menus or may launch a separate window for GUI input. Releasing the mouse button while a menu choice is highlighted causes that menu function to activate. Additionally, each drop-down menu has a "tear-off" feature which can be used to keep the menu on screen in a separate window without the need to continually re-pick the menu button. The drop down menu is torn off by selecting the dotted line at the top of the drop down menu.

The menu bar can be expanded and collapsed to reveal subsections via two controls at the lower left edge of the menu bar window (narrow buttons with triangles). One subsection is a toolbar, which is shown by default. The toolbar has buttons for often-used functions such as opening a file, object deletion, changing and resetting the view, and text command entry. A short description of what the button does can be shown by positioning the pointer over the button of interest.

The other control will expand the menu bar to reveal a text command entry box. While the user has access to all system functionality through the GUI, equivalent keyword-drive commands are available for all of this functionality. These commands are listed in Appendix A. The user is always free to enter such keyword-driven commands directly in the "Text Command" entry field on the main menu. Also, by clicking on the "Command Line Mode" button, the user will exit the graphical interface and be placed in a purely command line driven textual mode. The graphic interface can then be recovered by entering the `GRAPHIC` command at the command line prompt.

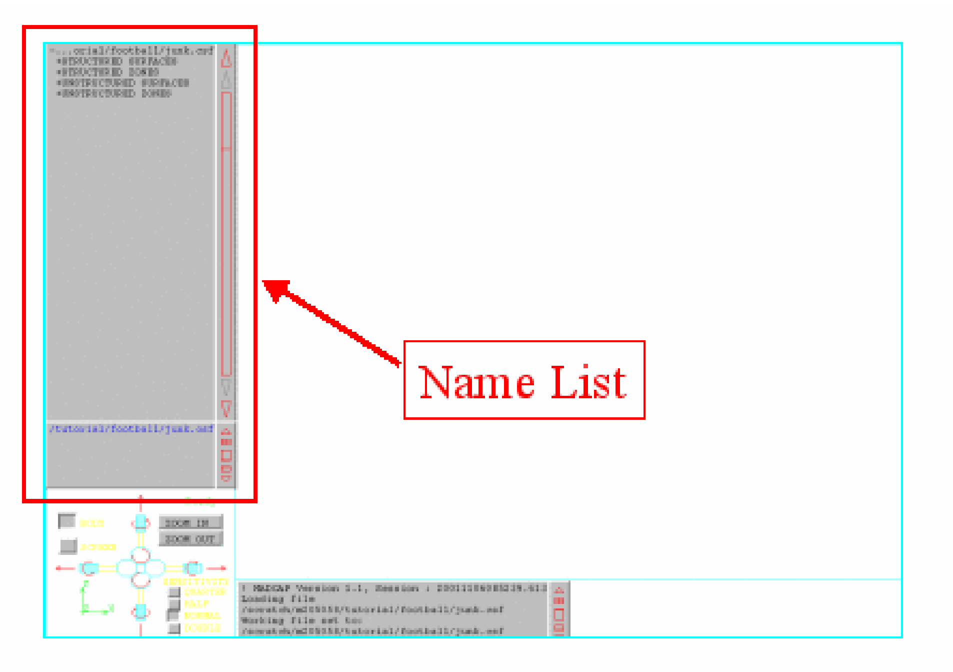
The initial graphics window for the MADCAP system, with no files loaded, is shown below.


Ready
BODY
SCREEN
ZOOM IN
ZOOM OUT
+Z
-Z
SENSITIVITY
S= 1.00000
0.5X
0.8X
1.25X
2.0X
! MADCAP Devel4.x 3Q2005, Session

2 Graphics Screen

The main Modular Aerodynamic Design Computational Analysis Process (MADCAP) graphics window is subdivided into three distinct areas. Each of these areas is described in detail below. Additionally, help is provided for the various peripheral input devices.

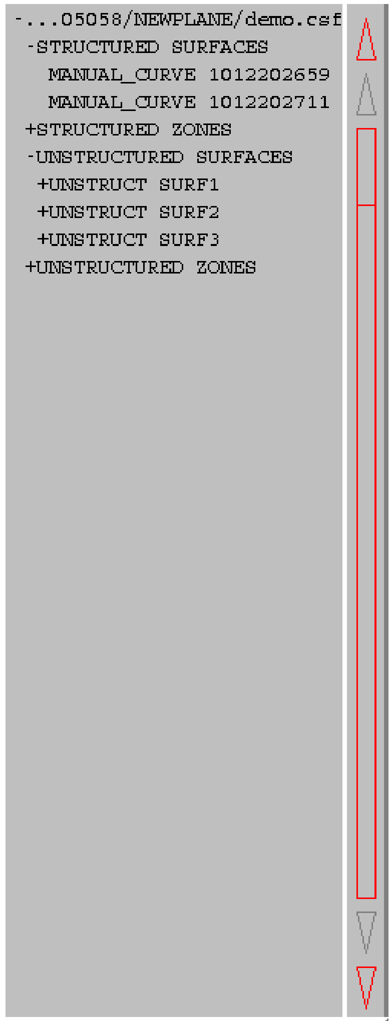
2.1 Name List



The graphics window name list is located on the left side of the main graphics window. The contents of the name list depends on the type of file that is currently set as the current display file.

2.1.1 General Information

A typical graphics name list (a *.csf* file here), is presented below.



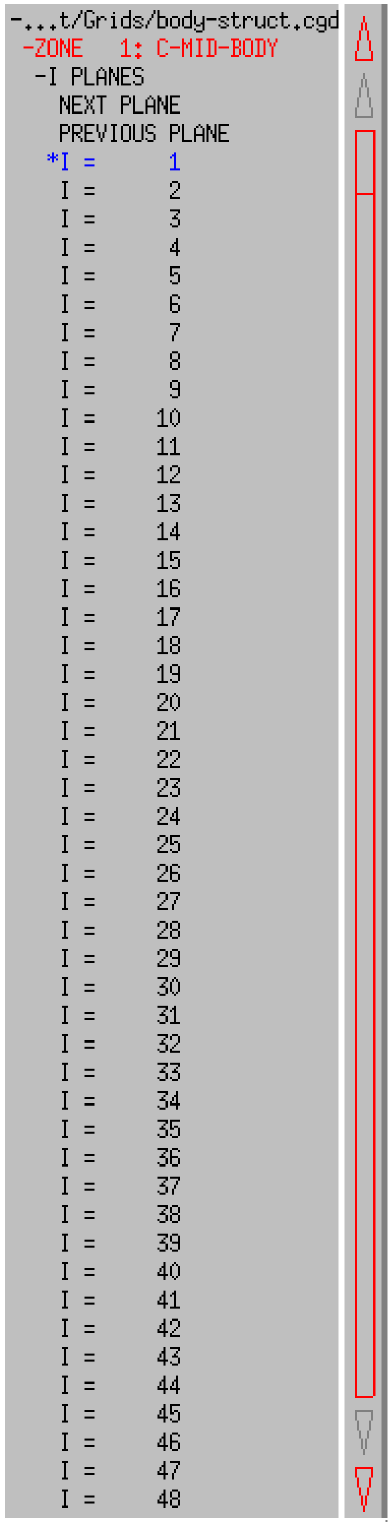
On the far right side of the name list are two up arrows, two down arrows, and a large vertical rectangle. These are buttons which can be used to control scrolling through the name list. Holding down the left mouse button on the lowest downward-pointing arrow will scroll the list forward one surface at a time. Clicking on the left mouse button on the other downward-pointing arrow scrolls the list forward one page at a time. Similarly, holding down the left mouse button on the highest upward-pointing arrow will scroll the list backward one surface at at time, while clicking on the other upward pointing arrow scrolls the list backward one page at a time. Clicking with the left mouse button anywhere within the rectangle will move the top of the display list to a point in the full list proportional to the vertical location of the pick in the rectangle. A horizontal line is drawn in the rectangle to indicate the relative location of the current top of the display list in the full list.

Some list items in the above example are seen to have a "+" or "-" next to them. These indicate list items which can be expanded or collapsed to show related groupings underneath. In this case, the `STRUCTURED SURFACES` item has been expanded, showing a list of structured surfaces contained in the grid file. List items may be expanded or collapsed by selecting them with the middle mouse button. Items may be shown on the graphics window by selecting them with the left mouse button. Selecting the item again will remove it from view. Selecting a group heading (such as `UNSTRUCTURED SURFACES`) with the left mouse button will display all of the objects underneath that heading.

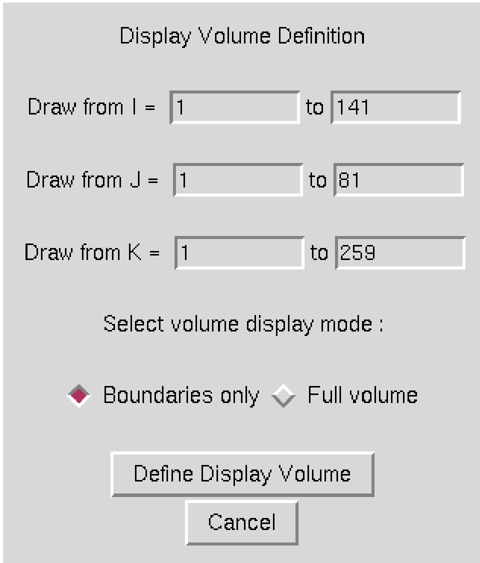
2.1.2 *.cgd* Files

If the Current Display File is a *.cgd* file, the initial name list presents the list of zones in the file. An example zone list in MADCAP is shown in Section 2.1.1.

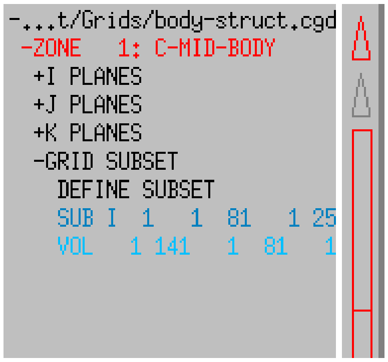
The top line of the name list always contains the name of file currently set as the Display File. The zones are presented in numerical order. To display planes from a zone, the name is picked from the list by selecting with the middle mouse button, scrolling if necessary. An intermediate list will be presented which gives the option of accessing a list of I-planes, J-planes, K-planes or grid subsets which may be defined. When I, J, or K planes are picked, that list item is expanded to present the list of planes, as seen below.



Any currently displayed planes are marked with an asterisk (*), and the names are drawn in the same color the plane itself is drawn with. To toggle a plane on or off, its name is picked from the list presented. To manually step through the grid planes, "Next Plane" and "Previous Plane" commands are available in the list. (See the Sweep Planes View menu option, described in Section 4.15 for a mechanism to sweep through grid planes automatically.) To define a subregion of a grid plane or a volume subset, select the "Grid Subset" list item. This will cause a new window to be displayed, as shown below.



A volume subset may be selected by entering the "from" and "to" indices for the desired region. A plane may be selected by entering identical "from" and "to" indices for the desired plane. The radiobuttons at the bottom of the window are used to set whether to display only the boundaries of the specified volume or the entire volume (every grid plane in every direction). Displaying the full volume of large grids is extremely graphics-intensive and should be used sparingly. After the desired subregion has been defined, the user should pick the "OK" button, and the specified volume will be displayed and added to the name list. An example of a portion of this list is shown below.

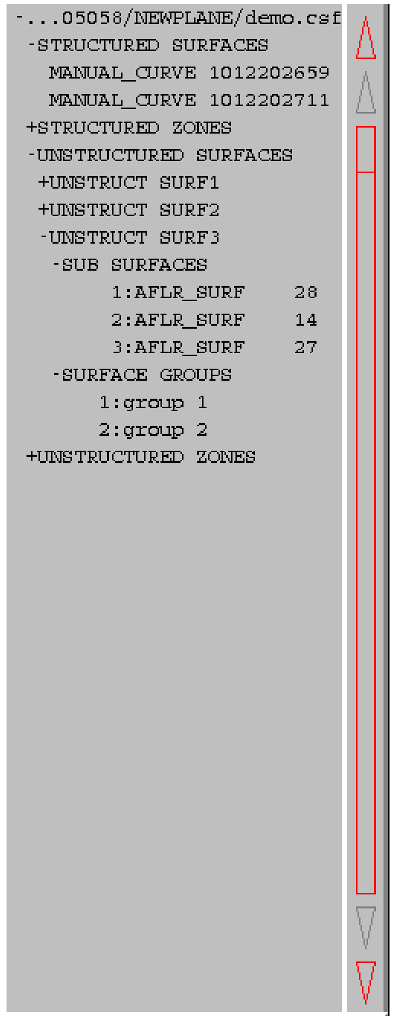


The `VOL` or `SUB` flag signifies whether the list entry is for a defined volume or subset. The indices following are the start and end indices of each computational direction on the volume or subset.

2.1.3 *.csf* Files

If the Current Display File is a Common Surface File (<i>.csf</i>), the name list contains the list of structured and unstructured surfaces and zones in the file. This list may be presented either alphabetically or chronologically depending on the setting of the Surface File Sort Mode flag in the View menus, as described in Section 4.17. The list is separated into Structured Surfaces, Structured Zones, Unstructured Surfaces, and Unstructured Zones. Selecting one of those items with the middle mouse button will expand the list to show the list of items of that type.

If a surface is picked from the list with the left mouse button, it is displayed on the main screen, and the name is drawn in the list in the same color as the displayed surface, and with an asterisk (`*`) preceding the name in the list. Selecting a zone name with the middle mouse button expands the list to show the grid planes contained in the file under the picked zone name. Selecting the zone name with the middle mouse button again will re-compress the zone list. An example of this is shown in Section 2.1.2 above. Unstructured surfaces that contain sub-surfaces or surface groups can also be expanded to list the names of the sub-surface and surface groups by selecting the unstructured surface name with the middle mouse button. An example of this is shown below. Here the `UNSTRUCT SURF3` unstructured surface has been expanded to see that it contains three sub-surfaces and two surface groups. The entire unstructured surface can be display by selecting the surface name with the left mouse button. A single sub-surface can be displayed by selecting the sub-surface name with the left mouse button (for instance `1:AFLR_SURF 28`). Similarly a surface group can be displayed by selecting the surface group name with the left mouse button. All of the sub-surfaces can be displayed by left clicking on `SUB-SURFACES` and all of the surface groups can be displayed by left clicking on `SURFACE GROUPS`.

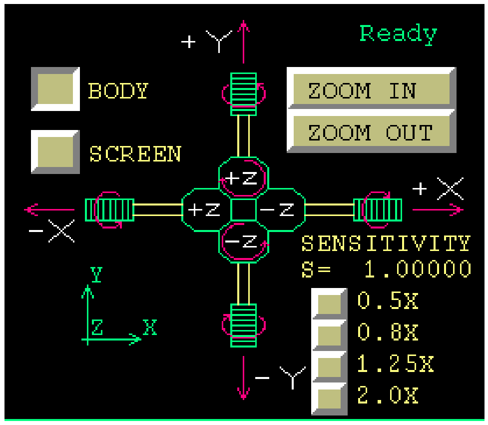


2.1.4 *.tmp* Files

If the Current Display File is a ZONI3G *.tmp* file, the name list contains the list of non-zonal surfaces existing in the file. Like in ZONI3G, these surfaces are listed in alphabetical order. Unlike ZONI3G, however, the MADCAP user has the option of displaying the list in chronological order using the Surface File Sort Mode menu, described in Section 4.17. Surfaces cannot currently be displayed from MADCAP. A conversion utility, described in Section 3.9.1, is available to convert ZONI3G *.tmp* files to the new Common Surface File (*.csf*), which is fully supported in MADCAP. The *.tmp* file list display may be useful in determining the contents of *.tmp* files while using the system's other features.

2.2 Screen Rotation and Translation Controls

On screen access to rotation, translation, and scaling is provided at the lower left of the main graphics window. This area is shown below.

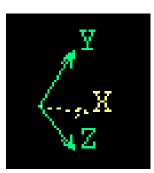


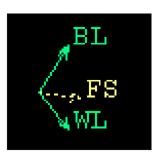
The control handle in the center of this area is used for all object rotation and translation. The arrows pointing out from the handle are used for x and y translations. Holding down the left mouse button on these arrows activates these translations. Similarly, the handles themselves perform positive and negative rotations about the x and y axes. For the z axes pointing in and out of the screen, four hexagonal buttons are provided. The rotation buttons are distinguished from the translation buttons by the circular arrow on the rotation buttons.

By default, rotations occur about the body axes. Holding down the translation or rotation handles described above rotates the geometry about the body x, y, or z axes. This typically correlates to rolling, pitching, and yawing the geometry. At times it is more desirable to rotate or translate the geometry relative to screen coordinates instead, to move the image up, down, left, or right, or to rotate the page. The `BODY` or `SCREEN` toggle in the upper left portion of this area is used to change the coordinate system the handle applies to. Note that the setting of this toggle also changes the sense of any dial and spaceball transformations, described in Section 2.5.

When the screen axes transformations are used, the z translation buttons cause the image to zoom in or out. Since this capability is almost always desirable even when the handle is set for body transformations, two buttons are provided in the upper right area of this screen to zoom in or zoom out when body transformations are set. Holding down the left mouse button on these buttons causes the image to zoom in or out. Either these buttons or the z translation buttons can be used to zoom the image when screen transformations are set.

A coordinate triad is displayed in the lower left portion of this screen area. The triad will have one of the following two forms depending on the setting of the Axis Mode menu option, described in Section 4.18.



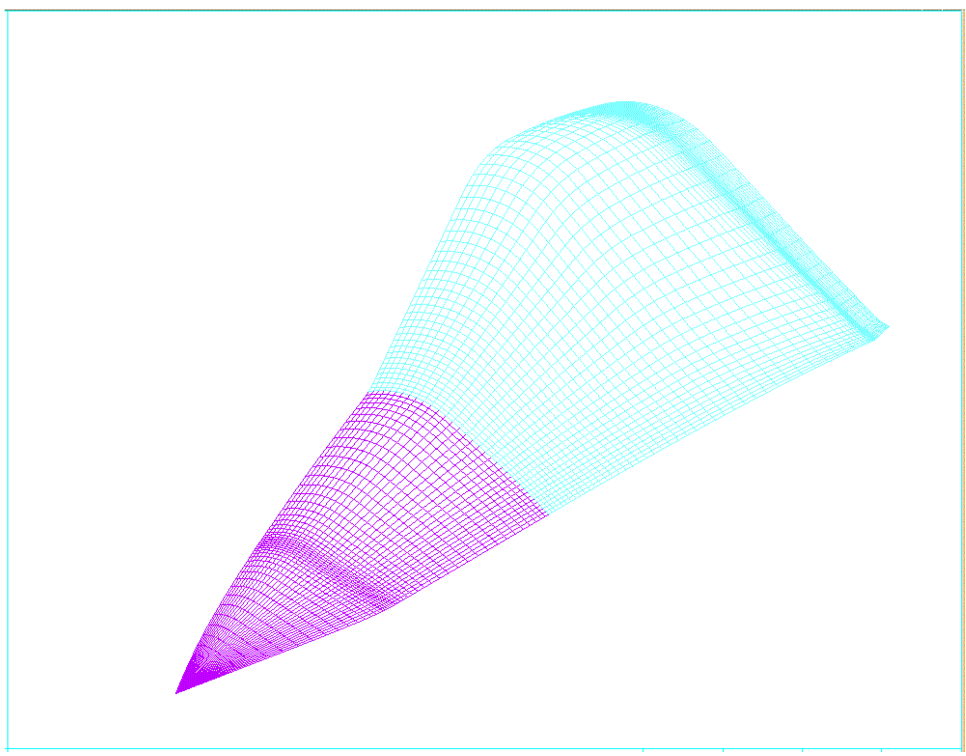


The image on the left shows the triad with `X`, `Y`, `Z` labels, while the one on the right shows the `FS`, `BL`, `WL` labels. Axes pointing into the screen are drawn in yellow in "Color on BG" mode, or as a dashed line when the color mode (see Section 4.13) is set to "FG on BG". The triad rotates with the geometry in real time as the geometry is rotated.

The lower right portion of the area contains a set of five toggle switches controlling the sensitivity of both the control handle and any dial and spaceball motion (see Section 2.5). There are five sensitivity levels to choose from: Normal, Double, Quad, Half, and Quarter. The default upon entry to the program is "Normal" speed, and the names of the other toggles indicate the speed relative to this default speed. The toggles are accomplished using the left mouse button.

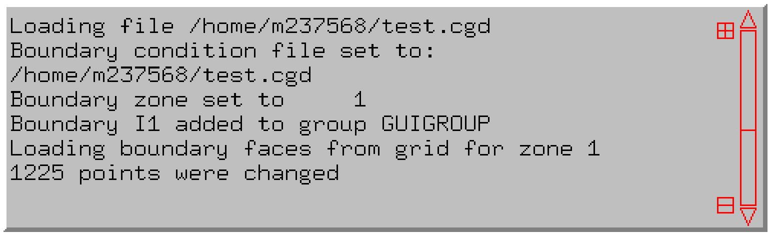
2.3 Main Display Area

The main display area is the large region of the upper right of the main graphics window. All objects requested for display are drawn in this region. An example wireframe geometry drawn in the display area is shown below.



2.4 Message Area

Some MADCAP operations cause informational messages to scroll to the message box located at the bottom of the screen. The message box can be interactively resized by the user by selecting the + and − boxes to the right of the message box, to a maximum of 10 lines or a minimum of 2 lines. The message can be scrolled through by using the triangles and rectangle at the right of the screen, similar to the name list display described in Section 2.1. An example message box is shown below.



2.5 Dial Box and Spaceball

MADCAP supports the use of both dial box and Spaceball peripherals for rotating the graphic image (dialbox and spaceball peripherals are currently not supported in the Windows environment). The user toggles the dial box or Spaceball inputs between body-based and screen-based axes using the toggles in the on screen rotation and translation portion of the screen, described in Section 2.2. The dials are mapped as shown below for 2×4 or 3×3 dialbox arrangement.

2×4 DialBox Arrangement

Zoom	*unused*
X-Rotate	X-Translate
Y-Rotate	Y-Translate
Z-Rotate	Z-Translate

3×3 DialBox Arrangement

X-Rotate	X-Translate	Zoom
Y-Rotate	Y-Translate	*unused*
Z-Rotate	Z-Translate	*unused*

In the absence of dials or space ball, the on screen rotation and translation screen area (Section 2.2) or the mouse (Section 2.6) may be used to manipulate the geometry display.

2.6 Mouse Buttons

There are two mouse modes in MADCAP, transformation mode and regular mode. In mouse transformation mode, the mouse in conjunction with the left, middle and right mouse buttons may be used to manipulate the geometry display. In regular mode, the mouse buttons are used to make object selections, center the display and get information about objects. In addition, the mouse may be used to toggle objects on and off.

Pressing the "m" key on the keyboard will toggle between the two mouse modes. When in the transformation mode, a "Mouse Transformation Mode" banner will be displayed across the top of the screen. While in the mouse transformation mode, objects can not be picked from the graphics display window.

In transformation mode, holding a mouse button while dragging the mouse in the graphics display window has the following effects. The left mouse button controls rotation of the display, the middle mouse button controls translation of the display, and the right mouse button controls zoom of the display. The rotation and translation axis are governed by the `BODY` and `SCREEN` buttons selected in the on screen rotation and translation screen area, described in Section 2.2.

In the regular mouse mode, each mouse button in MADCAP has a unique function when used alone or while the SHIFT key is held down. These functions are:

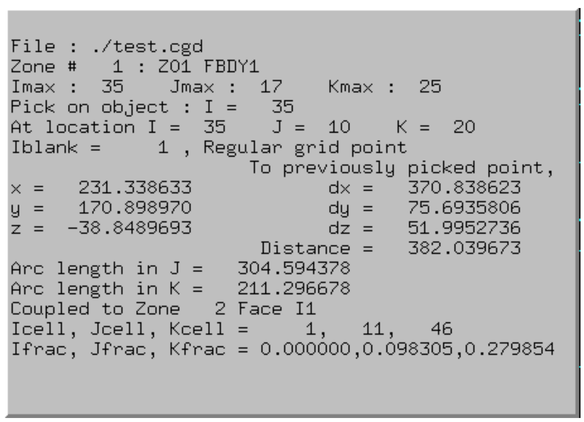
- Picking
- Centering

- Selecting
- Toggle Surface Off
- Toggle Surface On
- Turn Surface Off

2.6.1 Picking

Picking from the GUI, the graphic name list (section Section 2.1), and the on screen rotation and translation panel (Section 2.2) are all accomplished by using the left mouse button. These picks are typically performed on the downstroke, although the upstroke is used to signify the end of inputs such as list scrolling or stopping screen rotations in which the left mouse button is held down. If the left mouse button is used to pick on an object in the graphic display area, the program enters "Analysis" mode as described in the following paragraphs.

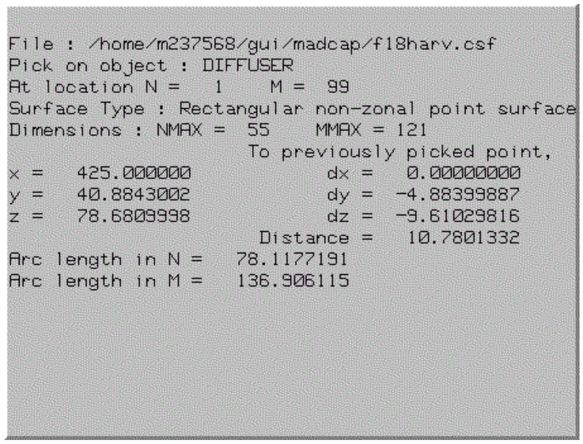
When the left mouse button is used to pick on a displayed object, an analysis box is drawn in the lower right corner of the display area. The type of information presented in the analysis box depends on the file type associated with the picked surface. Thus, an analysis pick on a *.cgd* file object will look somewhat different from an analysis pick on a *.csf* file object. Picking on a point on a *.cgd* surface with the right mouse button will cause the following type of window to pop up on screen, with an "`A`" drawn beside the picked point.



```
File : ./test.cgd
Zone #    1 : Z01 FBDY1
Imax :  35    Jmax :  17    Kmax :  25
Pick on object : I =   35
At location I =  35    J =  10    K =  20
Iblank =      1 , Regular grid point
                     To previously picked point,
x =    231.338633             dx =   370.838623
y =    170.898970             dy =   75.6935806
z =   -38.8489693             dz =   51.9952736
                        Distance =   382.039673
Arc length in J =   304.594378
Arc length in K =   211.296678
Coupled to Zone    2 Face I1
Icell, Jcell, Kcell =     1,    11,    46
Ifrac, Jfrac, Kfrac = 0.000000,0.098305,0.279854
```

The file associated is listed first, followed by the zone number, name, and dimensions of the zone that the picked object belongs to. The type of object picked (I-plane, J-plane, K-plane, Subset, or Volume) is output next. The computational indices of the picked point and its `IBLANK` value are presented next. The physical x, y, z coordinates of the point are output to the left, with the distance to the last point picked under analysis to the right. The total arc length of the displayed grid line in both computational directions is reported. (If a subset is selected, this is only the arc length of the subset, not of the full grid plane it is a subset of.) If the selected point has a boundary condition associated with it, either as an outer boundary or an overlapping fringe, this boundary condition is reported on the last lines, including the coupling parameters if it is a coupled boundary condition.

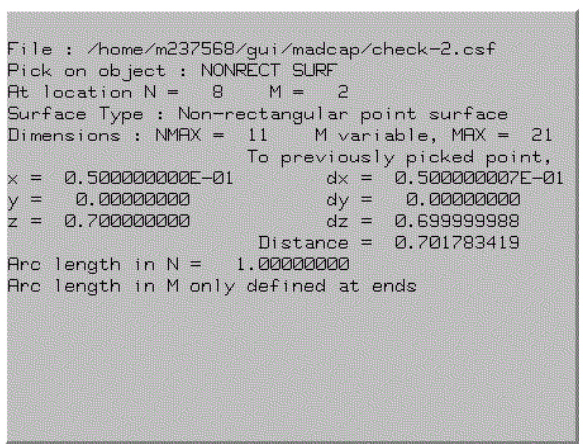
The contents of the analysis box when a pick is made on a `.csf` surface depends on the type of the surface picked. The general format of the analysis box is that shown below, for a pick on a two-dimensional, non-zone related surface.



```
File : /home/m237568/gui/madcap/f18harv.csf
Pick on object : DIFFUSER
At location N =    1     M =   99
Surface Type : Rectangular non-zonal point surface
Dimensions : NMAX =   55     MMAX = 121
                        To previously picked point,
x =    425.000000            dx =    0.00000000
y =    40.8843002            dy =   -4.88399887
z =    78.6809998            dz =   -9.61029816
                          Distance =   10.7801332
Arc length in N =   78.1177191
Arc length in M =   136.906115
```

The top line of the analysis box contains the name of the file associated with the picked surface. The next two lines contain the name of the object and the `N`, `M` location of the nearest node point picked on the object. The type of surface is displayed next, followed by the `NMAX`, `MMAX` dimensions of the surface. Coordinate data and distance to the previously picked point are displayed next. Finally, the arc lengths of the `N` and `M` curves at the picked point are presented.

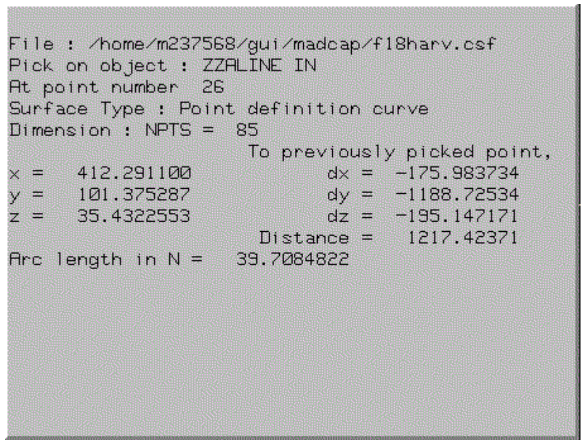
If the picked surface is non-rectangular, that is, all defining curves do not have the same number of points, the analysis box looks like this:



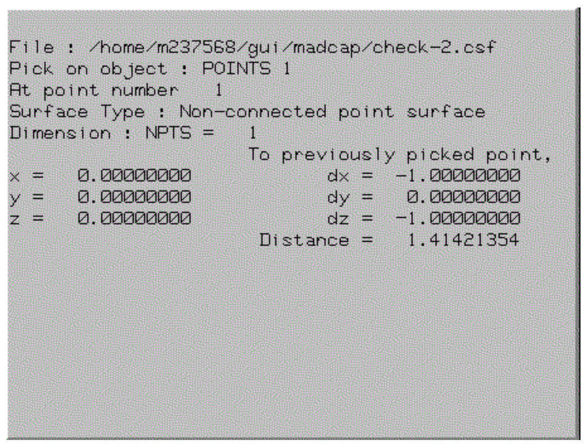
```
File : /home/m237568/gui/madcap/check-2.csf
Pick on object : NONRECT SURF
At location N =    8     M =    2
Surface Type : Non-rectangular point surface
Dimensions : NMAX =   11     M variable, MAX =  21
                        To previously picked point,
x =  0.500000000E-01         dx =  0.500000007E-01
y =   0.00000000             dy =   0.00000000
z =  0.700000000             dz =  0.699999988
                          Distance =  0.701783419
Arc length in N =    1.00000000
Arc length in M only defined at ends
```

Most data is the same as on the rectangular surface, but note that since the value of `M` is variable, this is written out and only the largest value of `M` is reported. Also, since the arc length in `M` can only be computed at the extremes in `N`, no value for arc length in `M` can be presented.

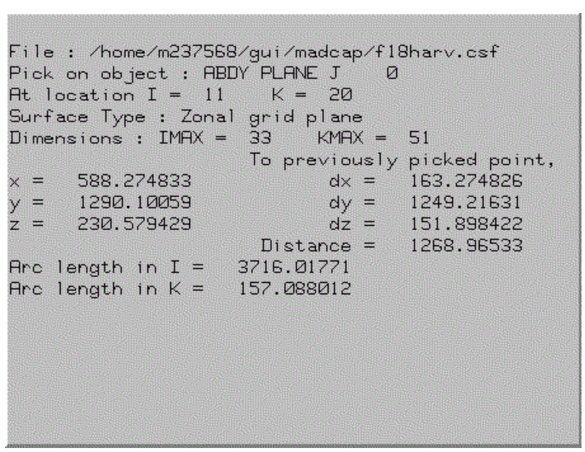
If the picked object is a single curve, the `M` data is removed completely, as shown in the example below.



If the picked object is a non-connected point type surface, no arc length data is available, and the display is as shown below.



Finally, if the picked surface is two-dimensional and rectangular, but is a zonal plane, the `N` and `M` descriptors are replaced with one of `I`, `J`, or `K`, as defined for the grid plane. An example is shown below.



2.6.2 Centering

The middle mouse button is used in MADCAP to re-center the graphic image. If the middle mouse button is pressed and released, the image is re-centered to the selected point on the display, with no rescaling being performed. If the middle mouse button is pressed and held down, a zoom box is swept out with the picked point at its center. When the middle mouse button is released after sweeping out a zoom box, the image is recentered to the selected point and rescaled such that the image inside the zoom box becomes the full display in the main drawing area, described in Section 2.3.

2.6.3 Selecting

The right mouse button is used to respond to prompts from the program to select displayed objects such as points, curves, surfaces, and zones. Depending on the type of object being prompted for, the right mouse pick will extract a lower dimensional object from any higher dimensional object. If the prompt is for "Any" object, the full object definition is selected. The right mouse button may also be used to select a name from the name list (see Section 2.1) if it is not displayed or hard to distinguish in the graphics display. If the prompt is for a point or curve and the user selects a two-dimensional surface name from the name list, the first point or curve in the surface definition will be selected. The right mouse button should also be used to select the on-screen buttons for aborting operations, signaling the end of multiple selection inputs, or selecting the contents of the object manager results buffer.

2.6.4 Toggle Surface Off

If the SHIFT key is held down while clicking the left mouse button over a displayed surface, the display of that surface is toggled off. Any surface whose display is toggled off in this way is saved in a reject buffer so that it can be quickly retrieved again by using the SHIFT-right mouse toggle on function described in the next section.

2.6.5 Toggle Surface On

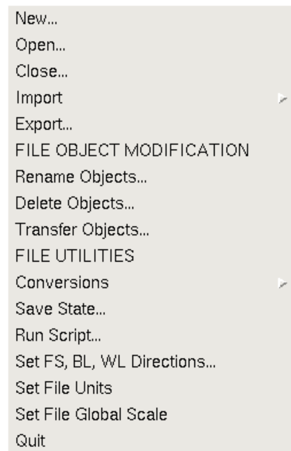
Any surface whose display has been turned off by using the SHIFT-left mouse toggle off function can be quickly retrieved by simply clicking the right mouse button in the main display area while holding down the SHIFT key. This function can continue to be used to retrieve previously toggled-off surfaces to the depth of the toggle reject stack, currently set at 100 surfaces. Note that when toggling surfaces back on, the currently set display file will be changed to the file associated with the surface you are toggling back on. Likewise, in a *.cgd* file, the currently set zone will be changed to the zone associated with the toggled on surface if it is not the zone currently set to display from in the *.cgd* display list (Section 2.1.2). Any surfaces that are associated with a file that has been closed (see Section 3.3) since it was first toggled off are removed from the reject stack and unavailable for re-display with this function.

2.6.6 Turn Surface Off

If the ALT key is held down while clicking the left mouse button over a displayed surface, the display of that surface is turned off. The surface will not be saved in the reject buffer and can not be redisplayed with the SHIFT-right mouse button.

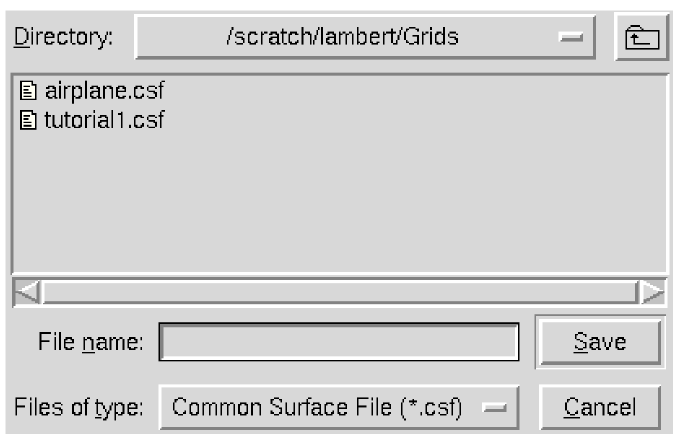
3 File Menu

Selection of the Modular Aerodynamic Design Computational Analysis Process (MADCAP) “File” menu item produces the drop down menu shown below.



3.1 New

Selection of the “New ...” submenu causes the following window to be displayed (note on Microsoft Windows OS platforms, the file dialog box may have a slightly different appearance).



This menu can only be used to open new files. Previously existing files can be opened via the “Open ...” menu, describe in Section 3.2.

The “Directory” dropdown menu at the top of the dialog allows the user to move up in the directory tree, as does the folder button next to it. The new file will be created in the directory that is shown.

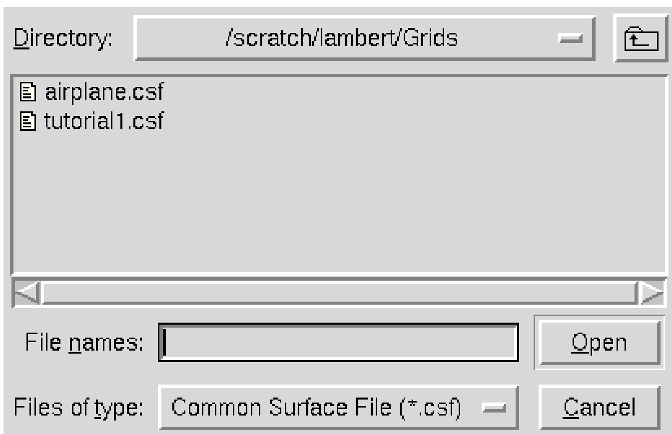
The drop-down menu at the bottom of the window is used to select the type of file to be created. The choices are:

.csf Common Surface Files
.cgd Common Grid Files

To open (create) a new file, enter the name of a new file in the "File name:" text entry box and click on the "Save" button with the left mouse button. The file name (including the file extension should be entered in the text box). The window closes and a new file with the selected name is created.

3.2 Open

Selection of the "Open ..." submenu causes the following window to be displayed (note on Microsoft Windows OS platforms, the file dialog box may have a slightly different appearance).



The "Directory" dropdown menu at the top of the dialog allows the user to move up in the directory tree, as does the folder button next to it. The drop-down menu at the bottom of the window is used to select the type of file to be opened. The choices include:

.csf Common Surface Files
.cgd Common Grid Files
.igs IGES Geometry Files
.tmp Surface Geometry (ZONI3G) Temp Files

Only the ".cgd", ".csf", and ".igs" options can display surfaces. It is not currently planned that the old *.tmp* file surfaces will be viewable in MADCAP, since that format is being replaced by the Common File *.csf* format. The list of surface names in *.tmp* files can be displayed, however, either alphabetically or chronologically. See the discussion of the "View / Surface File Sort Mode" sub-menu in Section 4.17 for details. *.cgd* files are the standard grid data file supported by Wind-US. These files typically are used to store the final volume mesh and contain one or more zone entities. The *.cgd* file usually contains the final boundary condition information for the flow solver and the coupling information between zones. *.csf* files (common surface file) is the MADCAP database format. This file contains all of the intermediate steps from a grid session. IGES files (see Section 3.2.1 contain geometry surfaces and curves. These files are static files in MADCAP (i.e., they can not be modified).

The selected file type menu not only controls the type of file to be opened, but also the types of files displayed in the file list. Only file names of the given extension, in the currently set Directory Path, are displayed. To list all files in the given directory, select "All Files(*)" from the file type menu. The directory path is set to the working directory from which MADCAP is being run. To change it, enter the new directory name in the entry box and enter a carriage return. To select the file to be opened, highlight its name in the list of files. If a new file is to be opened, set the directory path where the file is to be located and then type a new name in the entry box near the bottom of the window.

If the file being opened is a *.csf* or *.cgd* file, it is automatically set as the working file.

When the correct filename has been either selected or entered, click on the "Open" button at the bottom-right of the screen with the left mouse button. The window will close and the file will open. Multiple files may be selected by holding down the control key while selecting the names. Control will be returned to the MADCAP process, and all specified files will be opened, with the current fisplay file set to the last file specified to be opened. Currently, 10 files may be opened at any one time in the system.

To escape from this window with no action being taken, select the "Cancel" button.

The associated text command is:

`file open {tmp | cgd | csf | igs}` *filename*

3.2.1 IGES Files

MADCAP works on analytic geometry contained in IGES files. IGES files are static in MADCAP (i.e. they can not be modified). Only a subset of the full IGES specification is supported. The supported IGES entity types are listed in Table 1. IGES files containing other entity types, or geometry formats other than IGES files such as CATIA CAD files can be imported into MADCAP using the "Import / Geometry" menu, described in Section 3.4.2.

Requirements for IGES files are listed below.

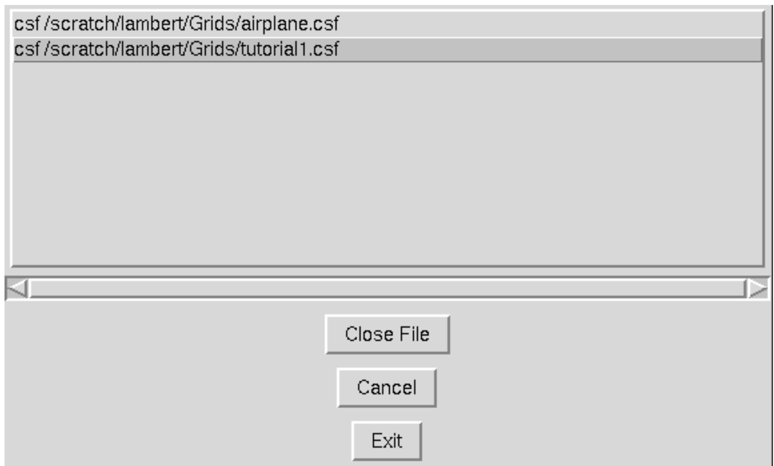
- Trim curves
  - Should include parametric curves.
  - Physical space curves are optional.
  - Boundary loops should be closed in parameter as well as physical space.
- Intersecting surfaces should be broken at the intersection curves.
- Avoid "excessive" number of surfaces.
- Surfaces should be broken at tangent discontinuities.

Table 1: Supported IGES File Entities

Number	Description
——— *Trimmed Surface Types*	
143/141	Bounded Surface/Boundaries
144/142	Trimmed Surface/Curve on a Parametric Surface
——— *Surface Types*	
108	Plane
114	Parametric Spline Surface
122	Tabulated Cylinder
128	Rational B-Spline Surface
120	Surface of Revolution (will be added by 3/03)
——— *Curve Types*	
100	Circular Arc
102	Composite Curve
104	Conic Arc
106 -11,-12,-13	Copious Data (representing a linear curve)
-63	Copious Data (representing a closed planar curve)
110	Line
112	Parametric Spline Curve
126	Rational B-Spline Curve
——— *Other*	
124	Transformation Matrix

3.3 Close

Selection of the "Close ..." submenu causes the following window to be displayed.



The files currently open in MADCAP, and their associated types, are listed at the top of this window. (Types are displayed to the left of the file name in the list.) To close any file, select a file name with the left mouse button, and then pick the "Close File" button. If no other files are to be closed, choose the "Exit" button. Otherwise, continue selecting file names and picking "Close File" until all files to be closed have been specified. When the "Exit" button has been picked, control will be returned to the MADCAP process, and all specified files will be closed. Any surfaces currently displayed from these files will be turned off. If the current display file is closed, the first remaining name in the list of open files will be set to the Display File.

To escape from this window with no action being taken, select the "Cancel" button.

The associated text command is :

`file close {tmp | cgd | csf | igs}` *filename*

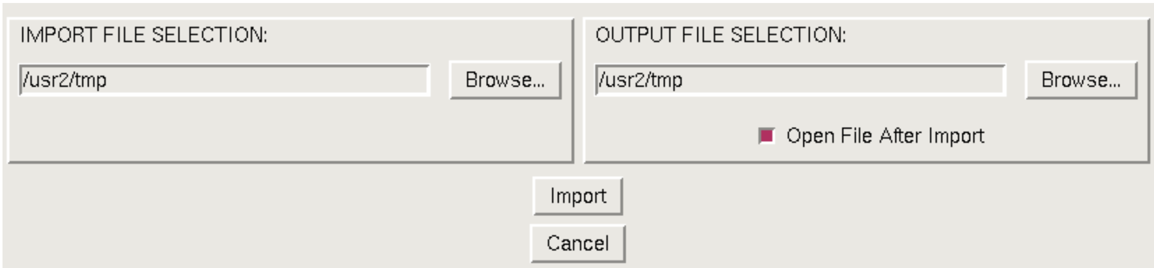
3.4 Import

Selection of the "Import" submenu causes the following pulldown menu to be displayed.

The "Import" menu is used to import data from grid or geometry files that are not in one of the MADCAP standard formats. Several file formats are supported.

3.4.1 Import Grid File

Selection of the "Grid File" menu option causes the following window to be displayed.

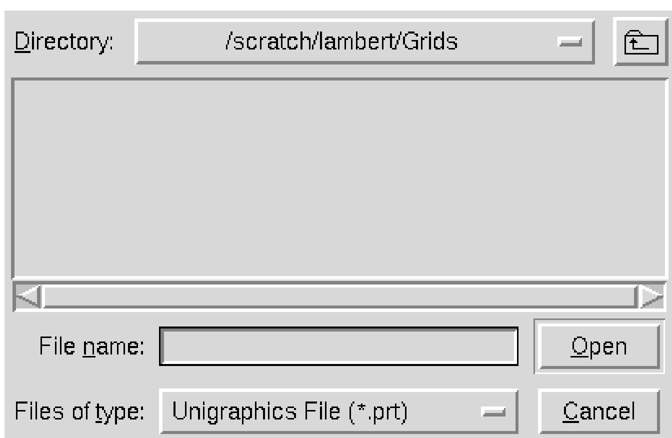


File formats other than the primary MADCAP formats (*.csf*, *.cgd*, *.igs*) can be imported into a *.csf*/*.cgd* file from this menu. The objects from the file can be imported directly into an open *.csf*/*.cgd* file or into a new or existing file. The file to be imported is selected on the left side of the panel, the file to import into is selected on the right side of the panel. The "Open File After Import" radio button can be selected to automatically have MADCAP open the *.csf*, *.cgd* file after file import.

Supported unstructured grid formats include AFLR (ugrid, sgrid, fgrid, etc), AFLR surface, fast, nastran, gambit, CGNS, and tecplot. In addition, structured grids from plot3d, temp, tecplot, agm, and hess files may be imported. Edge meshes from Fieldview particle paths can also be imported. The AFLR, AFLR surface, fgrid, fast, nastran, gambit, and CGNS formats use the AFLR file I/O library. The file names for these formats must follow the AFLR file naming convention.

3.4.2 Import Geometry File

Selection of the "Geometry File" menu option causes the following window to be displayed.

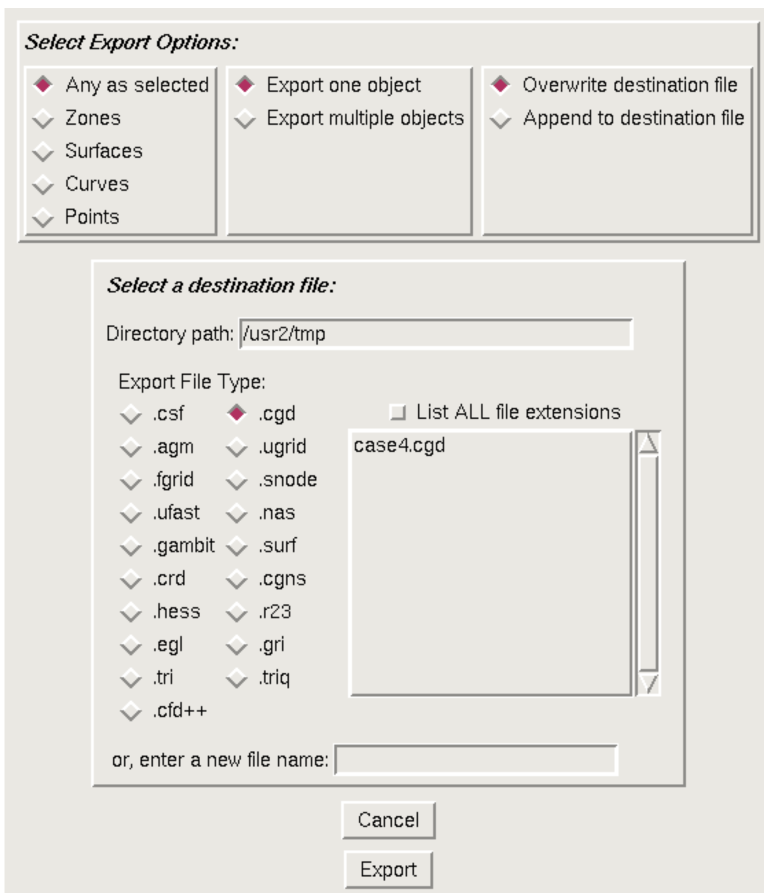


The geometry import feature provides a mechanism to import geometry models from STEP, IGES, Parasolid and CATIA v4 files. Use of this option requires an available CADFIX license on the machine on which you are running. In addition to the import of new geometry file types, the CADFIX import feature provides two additional benefits to MADCAP users. These are the ability to perform limited repair/preparation of geometry models and the ability to import geometry topology information from solid models. When you import a geometry into MADCAP through the CADfix import feature, the geometry will be prepared to be compatible with MADCAP geometry requirements. CADFIX will automatically perform operations such as conversion of unsupported entity types, and creation of parametric trimming curves for use by MADCAP. This reduces the potential for errors during the geometry import process and improves the overall robustness of getting complex geometry models into MADCAP. The CADFIX import feature will also allow you to read in the geometry topology information from your geometry model directly into MADCAP. This avoids the need to perform the costly boundary connections creation step in MADCAP. When you import a geometry file, a new IGES file with the extension *_cf.igs* will automatically be created and opened in MADCAP. In addition, a new *.csf* file with the same name will also be created and opened, and the geometry topology if available will be imported into a boundary data object in the *.csf* file.

The geometry file to be imported can be selected from the displayed list. The file type dropdown menu at the bottom of the window controls the files which are displayed.

3.5 Export

Selection of the "Export ..." submenu causes the following window to be displayed, and allows a user to transfer objects to other files.



The radio buttons at the top of the dialog control the type of objects to be exported, whether one or more objects are to be exported, and whether to overwrite the destination file.

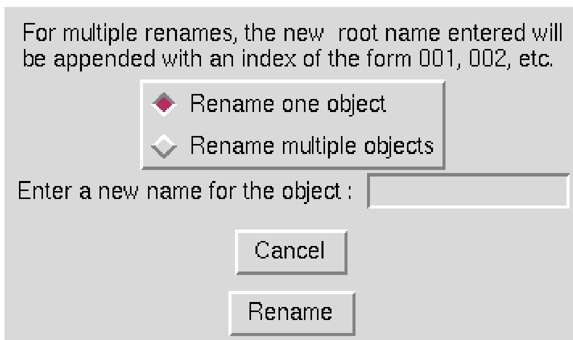
The lower part of the dialog contains details about the file to which the object or objects are to be exported. The directory path is where the file will be written, and defaults to the working directory. Several file types are available for conversion. Note that some file types do not support all object types. Some file types have additional options which will be displayed when the radio button for that type is displayed.

The fgrid, ufast, gambit, crd, ugrid, snode, nas, surf and CGNS formats use the AFLR file I/O library. The file names for these formats must follow the AFLR file naming convention. For example, ugrid files should contain the extension in the name and the file format will be determined by the extension name. For ugrid files these are *.r4.ugrid* for a binary single precision file, *.r8.ugrid* for a double precision binary file, etc. More information on AFLR file formats can be found in the AFLR user guide.

If exporting an object to an existing file, select that file in the file list. If exporting to a new file, enter the file name in the "new file name" text box.

3.6 Rename Objects

Selection of the "Rename Objects ..." submenu causes the window below to be displayed.



This option is used to rename any object in the current working file. Simply choose whether to rename a single object or a group of objects and enter a new name in the entry box. As indicated in the window, for multiple renames, the new object name consists of the root name entered appended with an index of the form 001, 002, etc. To begin the rename process, select the Rename button. The main graphical window will then prompt the user to select the object or group of objects to be renamed.

To escape this window with no action being taken, select the "Cancel" button.

The text commands generated are:

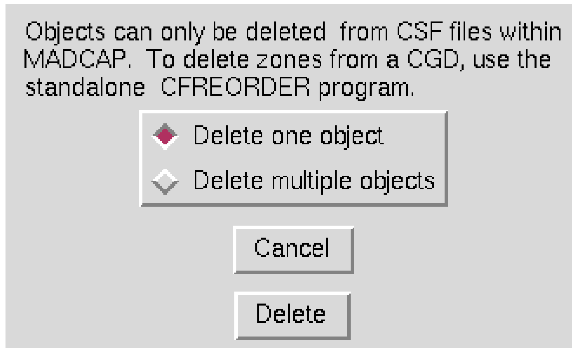
```
om select object_definition
file rename to newname
```

or

```
om looplist clear
om looplist object_definition (Multiple instances)
om loop begin
om select $list
file rename to newname
om loop end
```

3.7 Delete Objects

Selection of the "Delete Objects ..." submenu causes the window below to be displayed.



This option is used to delete objects in the current working file. Currently, delete only works for *.csf* files. To delete zones from *.cgd* files, the standalone program *cfreorder* must be used. In the future, this functionality will be included in MADCAP. To delete objects from a *.csf* file, simply choose whether to delete a single object or a group of objects and click on the Delete button. The main graphical window will then prompt the user to select the object or group of objects to be deleted.

To escape this window with no action being taken, select the "Cancel" button.

The text commands generated are:

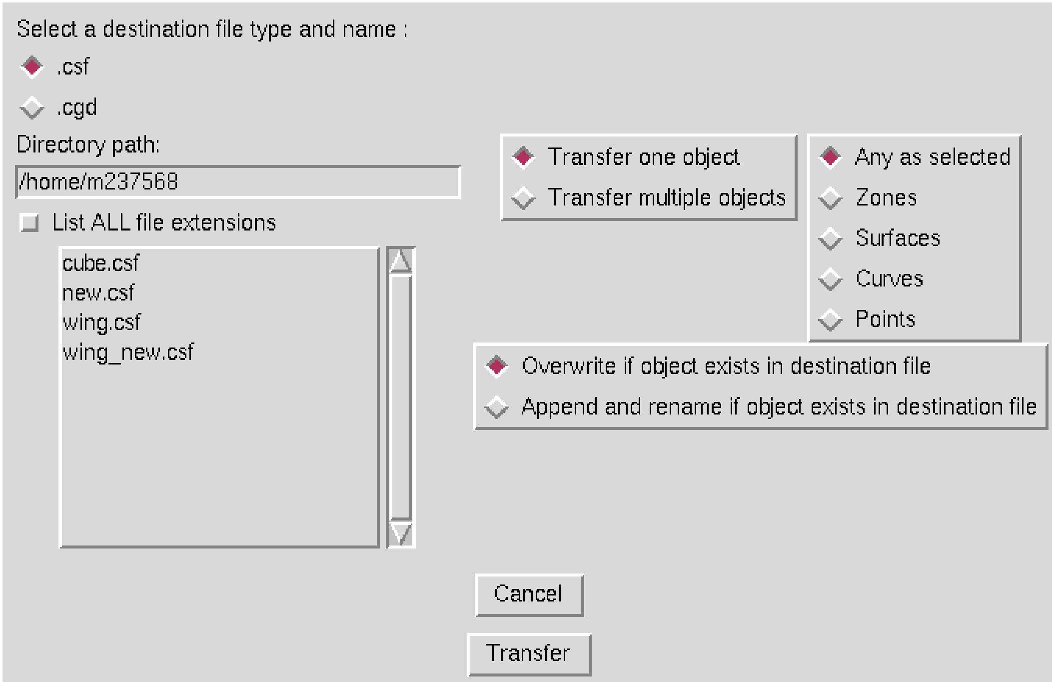
`om select` *object_definition*
`file delete`

or

`om looplist clear`
`om looplist` *object_definition* (Multiple instances)
`om loop begin`
`om select $list`
`file delete`
`om loop end`

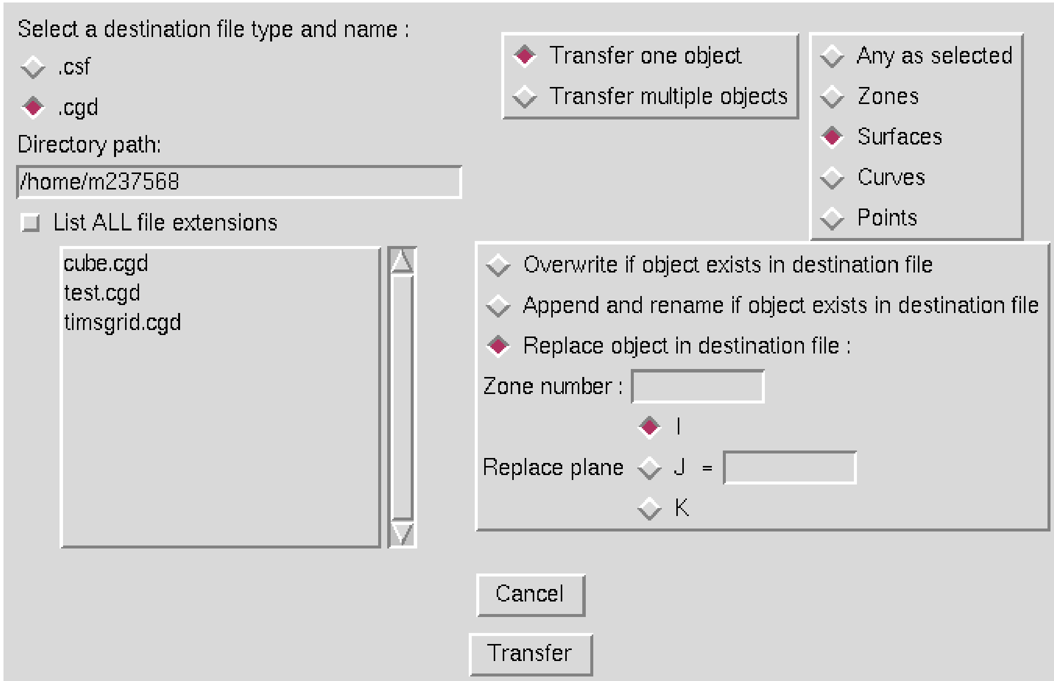
3.8 Transfer Objects

Selection of the "Transfer Objects ..." submenu causes the window controlling the transfer of object data between *.cgd* files and *.csf* files to be displayed. If the user sets the "destination file type" radiobutton to ".csf", the following window is displayed.



The user selects a *.csf* file to transfer to from the list of files presented. The directory path for these files may be changed as desired, and all file extensions may be viewed by setting the associated checkbutton. Three additional options should then be set to the right of the window. First, set whether a single object or multiple objects are to be transferred. Next, set the option for the type of objects to be transferred. If the "Any as selected" button is selected, objects of mixed type can be transferred, but the entire entity selected will be transferred. For example, you cannot transfer only a certain curve from a surface — the entire surface would be transferred. If the "Curves" option were set, any curve, existing as an independent object or a member of a surface or zone, could be transferred. Finally, the option for handling cases where an object of the given name already exists in the destination file must be set. This option can be set to "overwrite" or "append and rename".

If the destination is a *.cgd* file, the window presented is identical to that for a *.csf* file, as long as multiple objects are being transferred or the object type is set to "Any". Otherwise, an additional write option is presented for replacing a specific portion of a zone in the destination *.cgd* file, as shown in the window below for transferring a single surface to replace a specific plane in a specific zone of the destination *.cgd* file.



In this example, the user must enter the zone number, plane type, and plane index to be replaced when the selected surface is transferred to the *.cgd* file. In this manner, selected regions of zones in *.cgd* files may be modified by transferring smaller pieces of the zone from a different file. If the "Zones" object type is set, only the zone number to be replaced is entered. Similarly, if the option is "Curves" or "Points", the replace region is expanded to include the ability to specify a specific curve or point in the zone. Selecting the "Transfer" button will cause the MADCAP main graphical window to prompt for the objects to be transferred. The user should either select the appropriate objects or abort the operation using the on-screen abort button. When the final object has been selected, the objects will be transferred. To escape this window with no action being taken, select the "Cancel" button.

The text commands generated are:

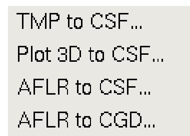
```
om select object_definition
file transfer to file [csf | cgd] filename [overwrite | \
   replace zone zonenum | insert zone zonenum ijkrange]
```

or

```
om looplist clear
om looplist object_definition (Multiple instances)
om loop begin
om select $list
file transfer to file [csf | cgd] filename [overwrite | \
   replace zone zonenum | insert zone zonenum ijkrange]
om loop end
```

3.9 Conversions

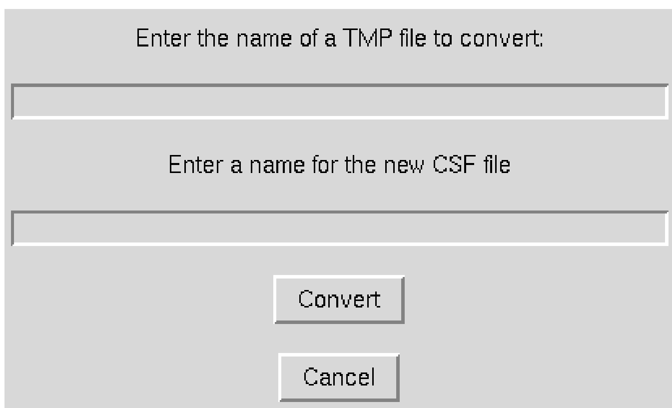
Selection of the "Conversions" submenu causes the following pulldown menu to be displayed.



This menu allow entire files to be converted from one format into the MADCAP *.csf* or *.cgd* formats. This capability is being replaced by the MADCAP "Import" menu, described in Section 3.4 which can also be used to convert file formats. Currently, conversions from the ZONI3G *.tmp* and PLOT3D formats to the Common Surface File (*.csf*) format are supported. Note, this menu is being phased out.

3.9.1 Temp to CSF Conversion

Selection of the "TMP to CSF ..." menu causes the following window to be displayed.



The name of the ZONI3G *.tmp* file to be converted is entered into the first entry area, and a name for a new Common Surface (*.csf*) File is entered in the second box. Selecting the "OK" button will cause the conversion process to begin. To escape with no action being taken, select the "Cancel" button.

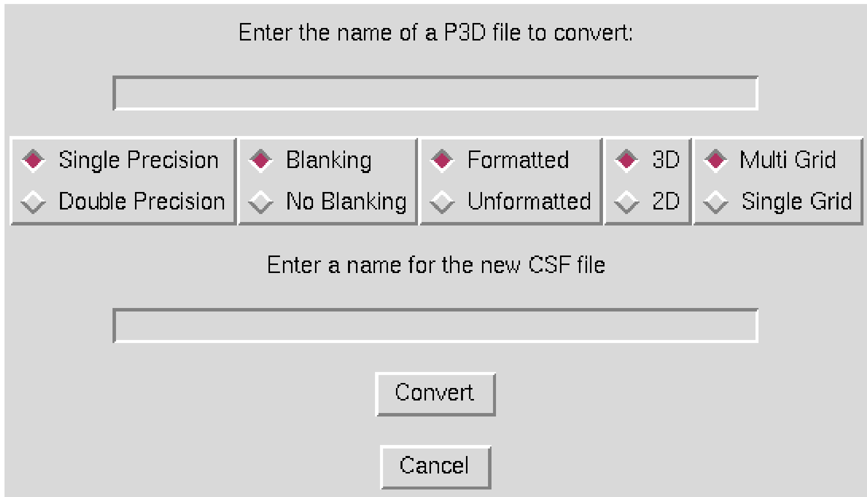
MADCAP messages are posted to the user as surfaces are transferred from the *.tmp* file to the *.csf* file. Currently, only ZONI3G type 106, 5001, and 7001 surfaces are supported in the *.csf* format. Thus, unstructured surfaces and zones are not transferred. Additionally, the zone node in the *.csf* format only supports grid planes, and not the zonal geometry or edge and corner surfaces available in the *.tmp* file. These surfaces are transferred into the STRUCTURED SURFACES node rather than into the ZONE node. The user is notified if any unsupported types are encountered in the conversion process, and if any Common File errors are generated during execution.

The associated text command is:

`file convert {tmp}` *filename* `{csf}` *filename*

3.9.2 PLOT3D to CSF Conversion

Selection of the "PLOT3D to CSF ..." menu causes the following window to be displayed.



The name of the PLOT3D file to be converted is entered into the entry box at the top of the window. All of the various PLOT3D file options must be set to match the attributes of the PLOT3D file being converted. Finally, a name for the *.csf* file to be created should be entered in the entry box at the bottom of the window. Selecting the "Convert" button will cause the conversion process to begin. To escape with no action being taken, select the "Cancel" button.

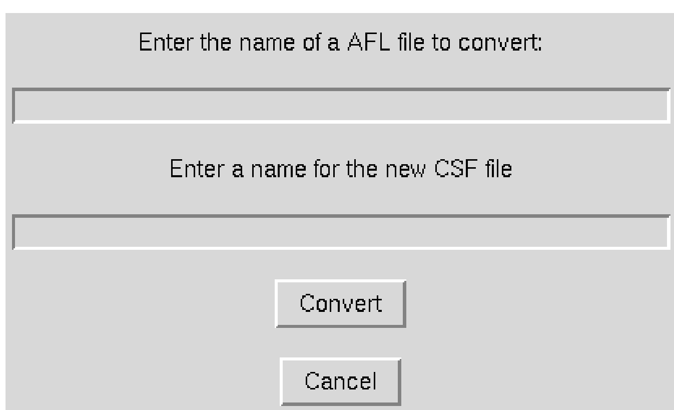
The associated text command (shown here on multiple lines for clarity) is:

```
file convert 'p3d filename [doubleprecision | standardprecision] \
              [blank | noblank] [formatted | unformatted] [3d | 2d]  \
              [multigrid | singlegrid]' {csf} filename
```

Note that any options must be included in single quotes with the filename.

3.9.3 AFLR to CSF Conversion

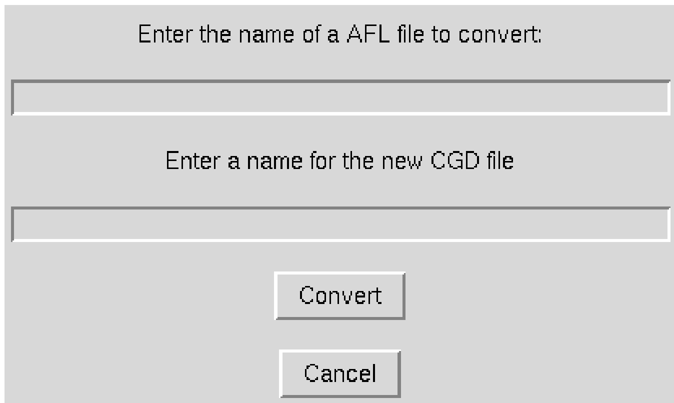
Selection of the "AFLR to CSF ..." menu causes the following window to be displayed.



The name of the AFLR file to be converted is entered into the entry box at the top of the window. A name for the *.csf* file to be created should be entered in the entry box at the bottom of the window. Selecting the "Convert" button will cause the conversion process to begin. To escape with no action being taken, select the "Cancel" button.

3.9.4 AFLR to CGD Conversion

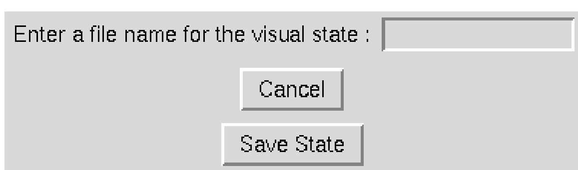
Selection of the "AFLR to CGD ..." menu causes the following window to be displayed.



The name of the AFLR file to be converted is entered into the entry box at the top of the window. A name for the *.cgd* file to be created should be entered in the entry box at the bottom of the window. Selecting the "Convert" button will cause the conversion process to begin. To escape with no action being taken, select the "Cancel" button.

3.10 Save State

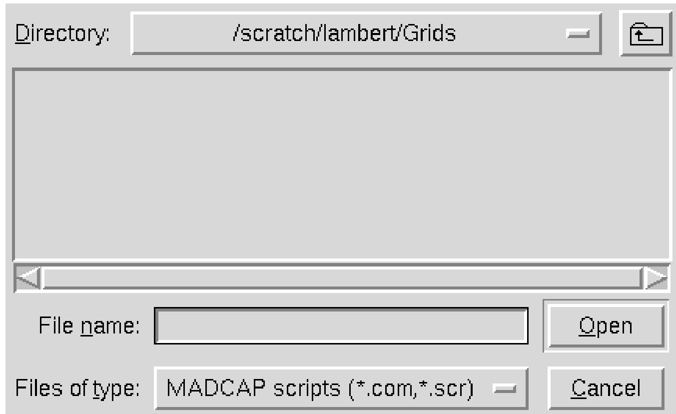
Selection of the "Save State ..." submenu causes the following dialog to appear.



Enter a file name to save the current state in. The current open files will be saved along with currently displayed objects and the current view. This file can be run as a MADCAP script in a future session to restore the current state. To execute the script, type "script *filename*" in the "Text Command" entry field on the main menu.

3.11 Run Script

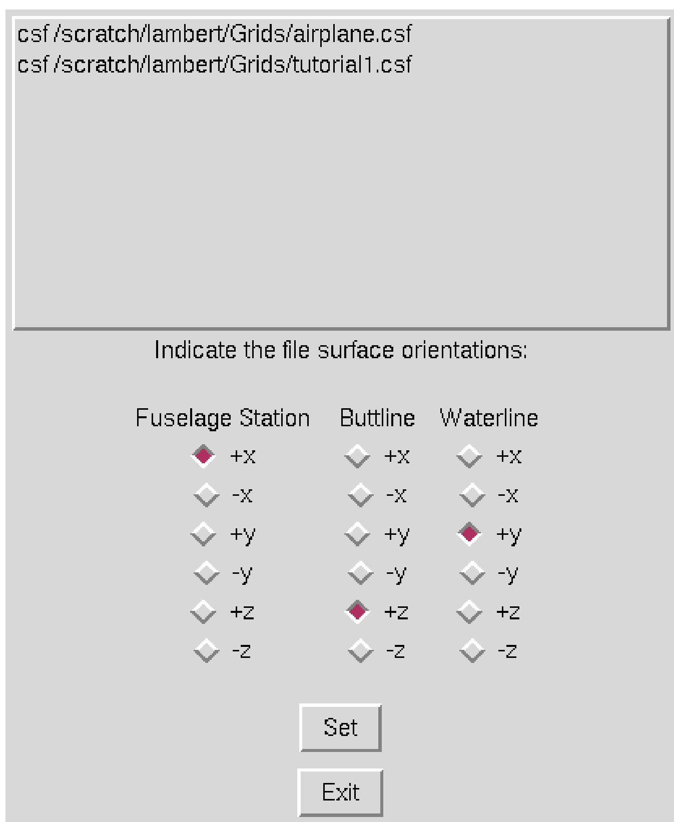
Selection of the "Run Script ..." submenu causes the following dialog to appear.



This dialog allows the user to execute a MADCAP script. The script name may be entered in the text box, or may be selected from the displayed file list. The "Directory" menu allows for navigating to another location to find the desired file. After the script is selected, selecting "Open" causes the script to be read and executed.

3.12 Set FS,BL,WL Directions

Selection of the "Set FS,BL,WL Directions ..." submenu causes the following window to be displayed.



MADCAP users have the option of displaying data in the pure x, y, z coordinate system in which the data has been stored, or in a fuselage station (FS), buttline (BL), waterline (WL) coordinate system. This window is used to relate the physical x, y, and z coordinates in the file with FS, BL, and WL directions. Different files may have different relationships between pure x, y, z coordinates and FS,BL,WL coordinates. In this manner, a file in which the up direction is $+y$ and a second file in which the up direction is $+z$ can be viewed consistently by setting the appropriate FS, BL, WL directions in this window, and then setting the View Axis Mode to FS, BL, WL, as described in Section 4.18. This control affects viewing of the data only — the coordinate values in the file are not changed.

The files currently open in MADCAP, and their associated types, are listed at the top of this window. To establish a relationship between x, y, z and FS,BL,WL for a given file, highlight a file in the list by selecting it with the left mouse button, and then use the three columns of radiobuttons to define the positive FS, BL, and WL directions. When the correct settings have been made, pick the "OK" button.

The default settings for a file upon opening are FS $= +x$, BL $= +z$, and WL $= +y$.

Setting the relationships in this window does *not* change the axis mode to FS, BL, WL, but only establishes relationships. To switch to FS, BL, WL axis mode, use the View Axis Mode menu item, described in Section 4.18.

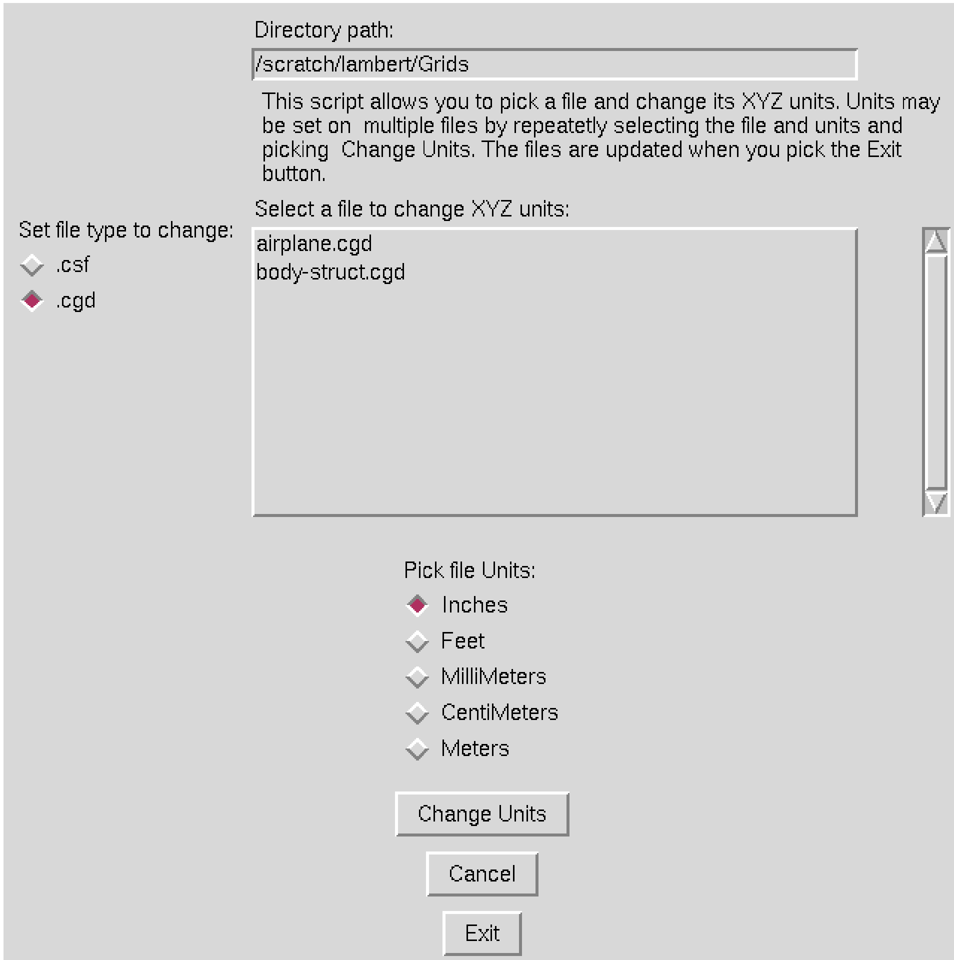
To leave this menu, choose the "Exit" button.

The associated text command is:

`file axisdef {tmp | cgd | csf}` *filename fsdir bldir wldir*

3.13 Set File Units

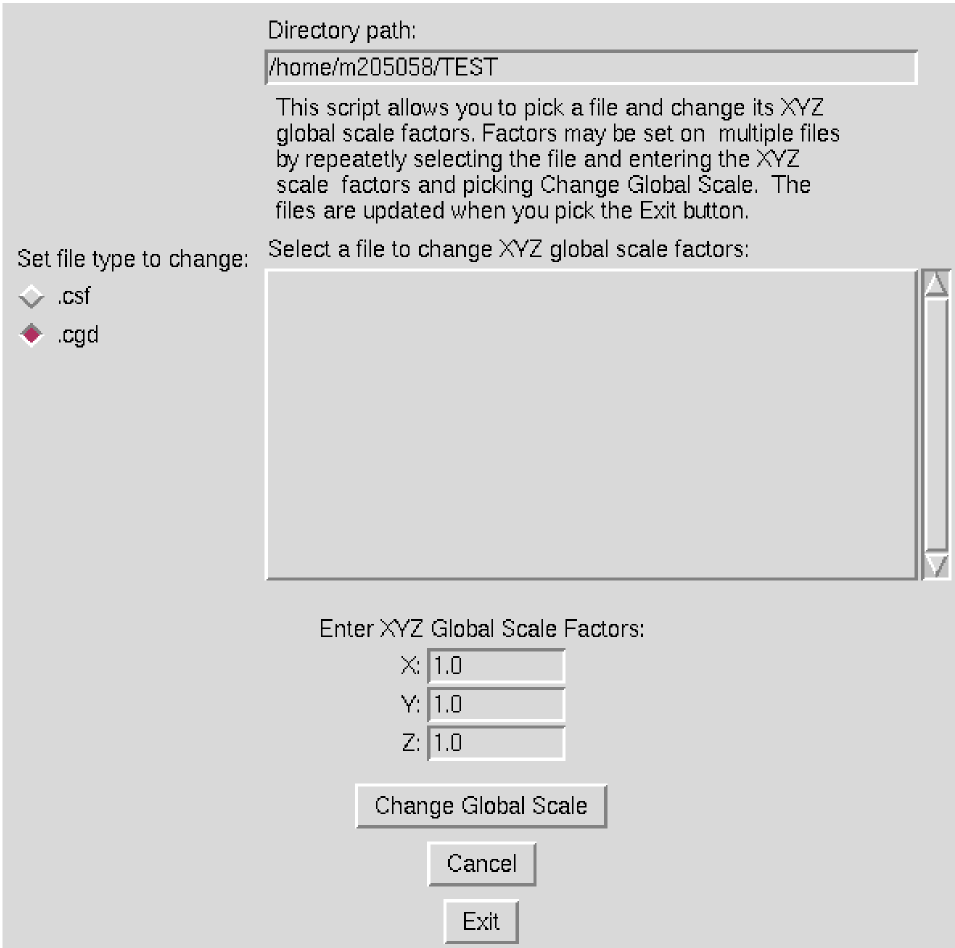
Selection of the "Set File Units" submenu causes the following to be displayed.



The units of a *.csf* or *.cgd* file may be changed with this menu. Select the file to be changed, and select the new units. The "Change Units" button will modify the file. To leave this menu without changing the file, select the "Cancel" button.

3.14 Set File Global Scale

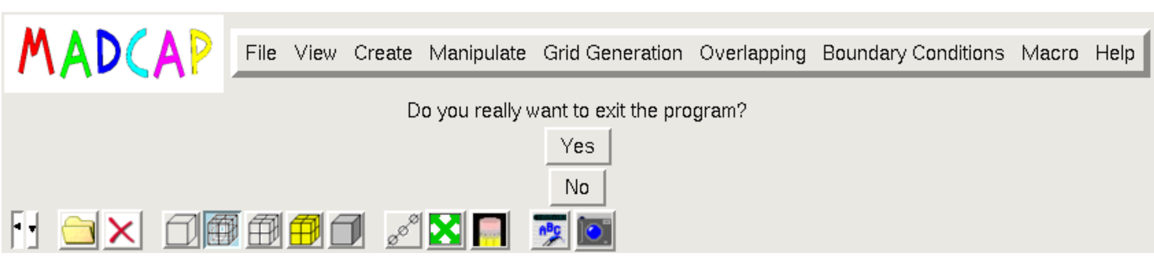
Selection of the "Set File Global Scale" submenu causes the following to be displayed.



The global scale factor of a *.csf* or *.cgd* file may be changed with this menu. Select the file to be changed, and select the x, y, and z scale factors. The "Change Global Scale" button will modify the file. To leave this menu without changing the file, select the "Cancel" button.

3.15 Quit

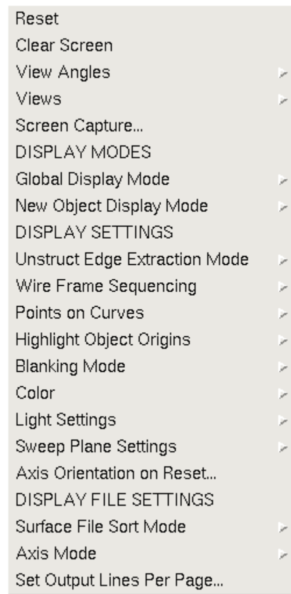
Selection of the "Quit" submenu causes the main GUI screen to change to look like this:



If "Quit" was chosen by mistake, choose the "No" button and no action will be taken — the system remains running at the point at which it was before the "Quit" button was selected. If you do intend to quit the program, you are given the option of saving the current program settings. Selecting the "Yes, and save settings" button will cause a file to be created in your working directory called *madcap*-your-user-id.*config*, which contains the settings in effect during this session for many of the MADCAP windows. The next time you run MADCAP from this directory, these settings will be used as default values in the MADCAP GUI. If you do not wish to create this file, select the "Yes, but do NOT save settings" button, and the program will terminate without the file being created. The next time you run MADCAP from this directory, the GUI will use all default values, or a previously generated configuration file.

4 View Menu

Selection of the Modular Aerodynamic Design Computational Analysis Process (MADCAP) “View” menu item produces the drop down menu shown below.



4.1 Reset

Selection of the “Reset” submenu causes the main graphics display to be resized to fill the main graphics window, at the default axis orientation (Section 4.16).

The associated text command is:

```
view reset
```

4.2 Clear Screen

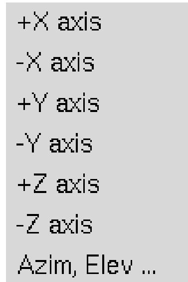
Selection of the “Clear Screen” submenu causes the main graphics display to be cleared of all displayed objects. All data is erased, but the viewing transformation remains unchanged. The graphics scale factor will be determined by sizing the next displayed object to fill the graphics window.

The associated text command is:

```
view clear
```

4.3 View Angles

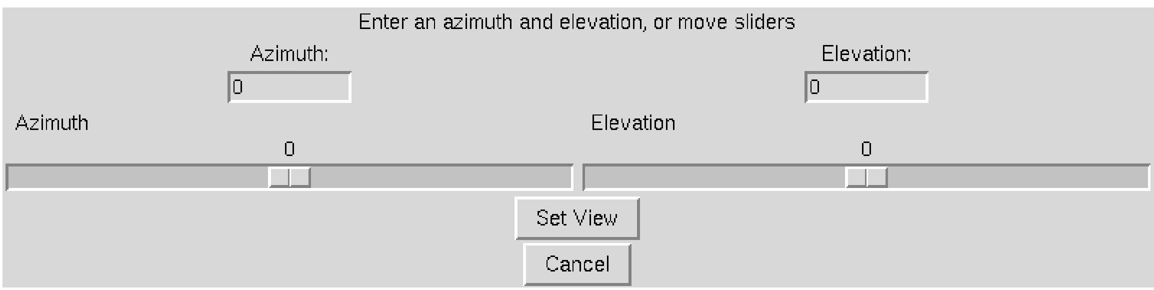
Selection of the “View Angles” submenu causes the following pullout menu to be displayed.



This menu option is used to specify a viewing direction either along one of the principal axes or by providing azimuth and elevation viewing angles.

If a direction along one of the principal axes is selected, the graphics display will be modified such that the selected axis points into the screen. The screen direction for the remaining positive axes will depend on the default (reset) axis orientation, as set by the Axis Orientation on Reset function described in Section 4.16.

If the "Azim, Elev" menu item is selected, the following window is displayed.



The user may either type elevation and azimuth angles to move the viewpoint to, or move the sliders to modify the values. These viewpoint transformations are relative to the default (reset) axis orientation, as set by the Axis Orientation on Reset function (Section 4.16). The azimuthal rotation is performed first, followed by the elevation rotation. Remember that these angles are rotating the viewpoint, not the geometry itself.

Releasing the mouse over the desired principal axis, or choosing the "OK" button on the elevation, azimuth input window will cause the display to be redrawn in the specified orientation.

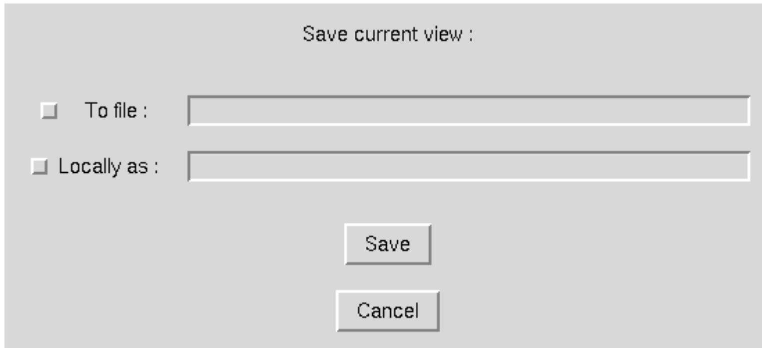
The associated text command is:

`view angle {+xaxis | -xaxis | +yaxis |-yaxis | +zaxis | -zaxis | \`
`azimelev` *azimuth elevation*`}`

4.4 Views

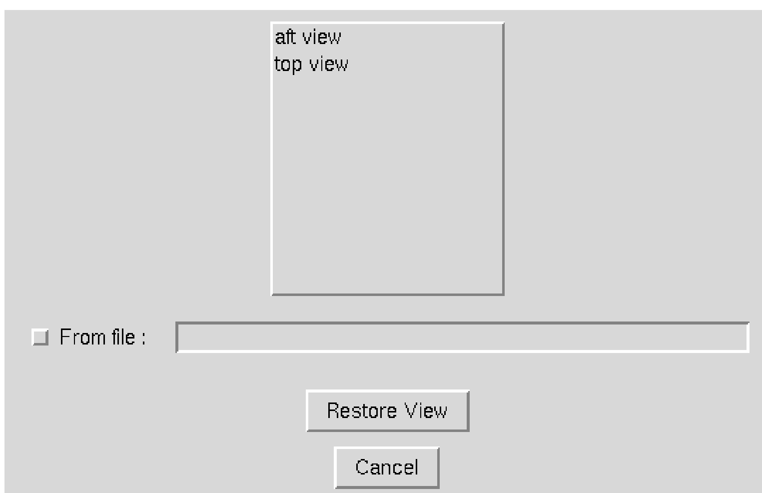
Selection of the "Views" submenu causes the following pulldown menu to be displayed.

If a viewing transformation has been created which the user may desire to use again in the future, it may be saved by selecting the “Save ...” menu option. This causes the following window to be displayed.



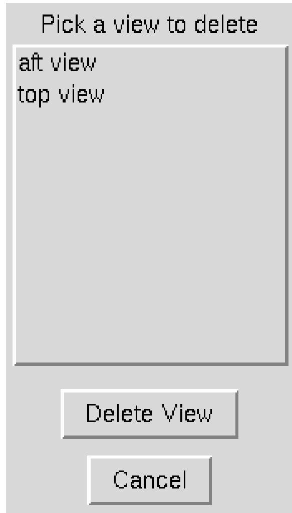
Views may be saved either to file or within a MADCAP session as a named view, or both. To save to file, the user must select the checkbutton labeled “To file”, and enter the name of a file to store the transformation in in the entry box. To save locally, select the checkbutton labeled “Locally as”, and enter a name for the view in the entry box. A saved view only contains the viewing transformation matrix — it does not maintain a list of displayed surfaces or display modes. Selecting the “OK” button will cause the current view to be saved. To escape with no action being taken, select the “Cancel” button.

If the “Restore ...” menu option is selected, the following window is displayed.



The list box at the top of this window contains the names of all views saved locally within this MADCAP session. To restore a view, the user must either select a name from this list or highlight the “From file” check box and enter the name of a file containing a previously saved viewing matrix. After either selecting a local name or a file name, selecting the “Restore View” button will cause the graphics window to be redrawn with the specified viewing transformation. To exit with no action being taken, select the “Cancel” button.

If the “Delete ...” menu option is selected, the following window is displayed.



This window contains a list of all named views in a given MADCAP session. There is a finite limit (currently 20) to the number of views that may be saved locally within a given session. If this maximum is reached, and the user needs to save a new view, a previously saved view must be deleted. This can be done by selecting a view name from the displayed list in this window, and then picking the "Delete View" button. To exit with no action being taken, select the"Cancel" button.

The associated text commands are:

```
view save    {file filename | name viewname}
view restore {file filename | name viewname}
view delete name viewname
```

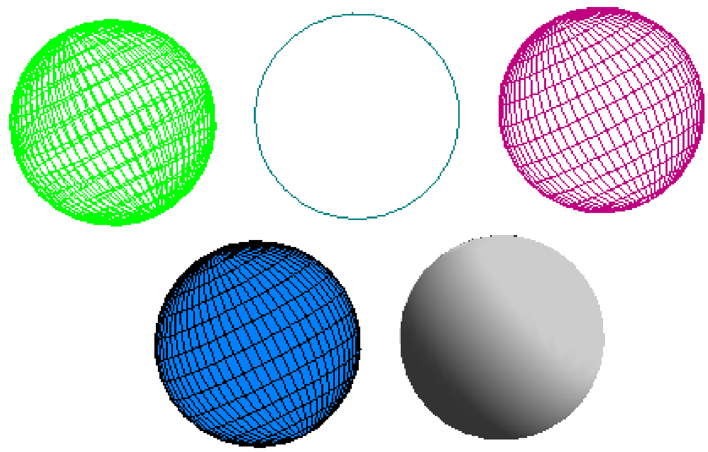
4.5 Screen Capture ...

4.6 Global Display Mode

Selection of the "Global Display Mode" submenu causes the following pulldown menu to be displayed.

The Global Display Mode determines the style in which all geometry displayed in the graphics window is drawn. The Global Display Mode overrides the Object Display Mode attribute which each drawn entity is assigned based on the current New Object Display Mode setting (Section 4.7). If the "As Generated" option is selected, the assigned display mode attribute is used, and each object is drawn in its own style. If any of the other options is selected, the entire display is drawn in that mode.

"Wireframe" mode draws objects as see-through wire meshes. "Edges Only" mode draws only the outlines of objects, and results in the fastest real-time rotations. The remaining modes, "Hidden Line", "Solid Panels", and "Lighted", all cause objects closer to the viewer to fully obscure objects behind them. In Hidden Line mode, the grid is drawn just as in Wireframe mode, but the cells are filled in the background color such that hidden lines are removed. In Solid Panels mode, the grid is drawn in the foreground color (typically white), and the cells are filled in the assigned surface color. In Lighted mode, no grid lines are seen and the object is drawn as a solid body illuminated by a light source. The color of the solid body is set under the Color submenu (Section 4.13), and the light source location is manipulated under the Light Settings submenu (Section 4.14). The images below show a sphere drawn in each of the possible display modes.



Releasing the left mouse button over the desired Global Display Mode will cause that radiobutton to be set, and the graphics display will be redrawn as specified.

The associated text command is:

```
view mode {asgenerated | wireframe | edges | hidden | solid | lighted}
```

4.7 New Object Display Mode

Selection of the "New Object Display Mode" submenu causes the following pulldown menu to be displayed.

The New Object Display Mode determines the style in which all future objects are displayed. Objects maintain an association with the display mode in which they are originally drawn. Thus, changing the New Object Display Mode has no effect on the current graphics display, but only on objects drawn after a selection has been made. By changing the New Object Display Mode, a mixture of drawing styles may be used at the same time in the display (as long as the "As Generated" option is set in the Global Display Mode, described in Section 4.6). The selected mode stays in effect until changed again. The initial default New Object Display Mode is Wireframe.

If new objects are being drawn in a mode other than the one set, check to ensure that the style is not being overridden by a Global Display Mode. Even if the Global Display Mode is set to something other than "As Generated", the object attribute will be assigned to the current "New Object Display Mode", not to the Global Display Mode.

"Wireframe" mode draws objects as see-through wire meshes. "Edges Only" mode draws only the outlines of objects, and results in the fastest real-time rotations. The remaining modes, "Hidden Line", "Solid Panels", and "Lighted", all cause objects closer to the viewer to fully obscure objects behind them. In Hidden Line mode, the grid is drawn just as in Wireframe mode, but the cells are filled in the background color such that hidden lines are removed. In Solid Panels mode, the grid is drawn in the foreground color (typically white), and the cells are filled in the assigned surface color. In Lighted mode, no grid lines are seen and the object is drawn as a solid body illuminated by a light source. The color of the solid body is set under the Color submenu (Section 4.13), and the light source location is manipulated under the Light Settings submenu (Section 4.14).

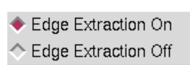
Releasing the left mouse button over the desired New Object Display Mode will cause that radiobutton to be set, and future objects will be drawn in that mode.

The associated text command is:

```
view objmode {wireframe | edges | hidden | solid | lighted}
```

4.8 Unstruct Edge Extraction Mode

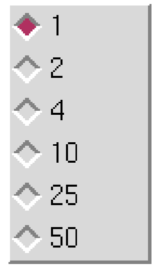
Selection of the "Unstruct Edge Extraction Mode" submenu causes the following pulldown menu to be displayed.



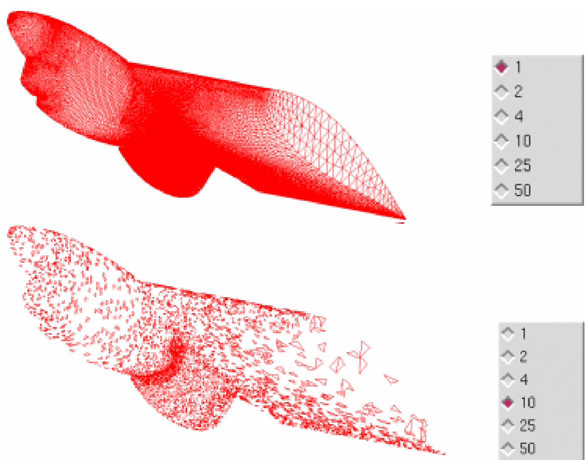
By default, every time an unstructured surface mesh is displayed, the free boundary edges of the surface are extracted to support the edges only display mode. For invalid surface meshes, the edge extraction operation can fail and cause the surface to not be displayed. This menu allows the edge extraction to be turned off during the display of future unstructured surfaces. If edge extraction is turned off, the edges only display will not be shown for unstructured surfaces. Releasing the left mouse button over the desired mode will cause that radiobutton to be set, and future unstructured objects will be drawn in that mode.

4.9 Wire Frame Sequencing

Selection of the "Wire Frame Sequencing" submenu causes the following pulldown menu to be displayed.



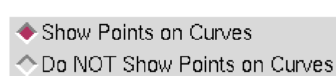
This menu allows the display of unstructured surface meshes to be sequenced by display a subset of the faces. The choices are to display every 1, 2, 4, 10, 25, or 50 faces. Note that this option only works for unstructured surface meshes. An example of the wire frame sequencing mode is illustrated in the following example.



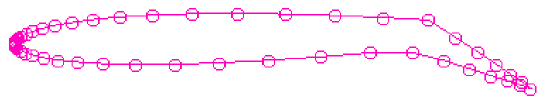
Releasing the left mouse button over the desired sequencing level will cause that radiobutton to be set, and future unstructured objects will be drawn in that mode. This mode can be useful to speed up the graphic display for objects containing a large number of faces. A typical use would be to set the wire-frame sequencing to a large number while the display is being manipulated (rotated, zoomed etc) and then set the wire-frame sequencing back to one once the final display view is set.

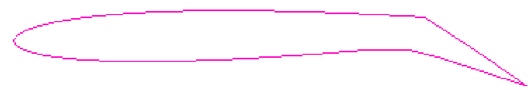
4.10 Points on Curves

Selection of the “Points on Curves” submenu causes the following pulldown menu to be displayed.



By default, objects consisting of a single curve in MADCAP are displayed with dots at the defined node points on the curve to assist the user in identifying them. This is usually the desired display mode when working in the program. However, there are times when it is desirable to have the dots not be displayed. These radiobuttons are used to control the display mode for individual curve objects. The images below show a typical curve displayed with and without points displayed.





Releasing the left mouse button over the desired option will cause that radiobutton to be set, and the screen will be redrawn such that all currently displayed curves reflect the current mode.

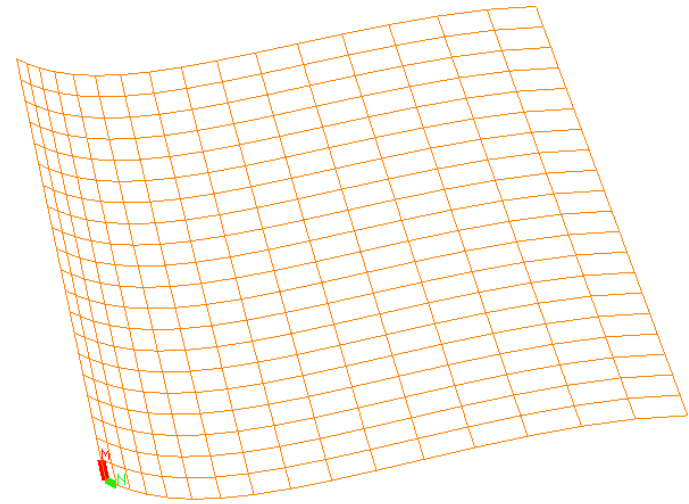
The associated text command is:

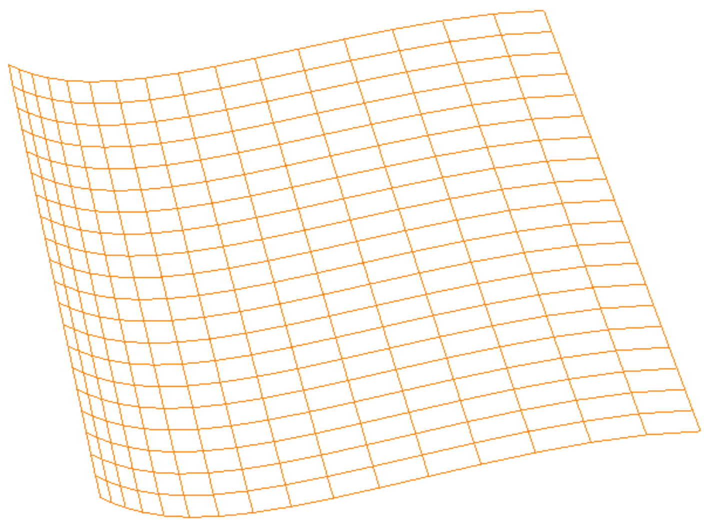
```
view curvepoints {on | off}
```

4.11 Highlight Object Origins

Selection of the "Highlight Object Origins" submenu causes the following pulldown menu to be displayed.

All objects in MADCAP are defined as point definition objects, with a definite point ordering scheme. Certain operations require the user to have knowledge of this ordering. For surfaces, the two families of grid lines are referred to as constant N lines and M lines. On constant N lines, the M index varies from 1 to the number of points on the N lines, and vice versa for constant M lines. To assist the user in identifying which lines are which, objects can be drawn with the first segment of the first N and M lines highlighted. The images below show a typical surface displayed with and without the surface origin highlighted.





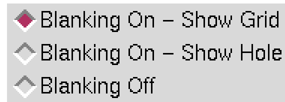
Releasing the left mouse button over the desired option will cause that radiobutton to be set, and the screen will be redrawn such that all currently displayed curves and surfaces are drawn with the object origin highlighted. When highlighted, the constant M = 1 line, on which N varies, is drawn with the first segment highlighted in green and a character "N" drawn at the N = 2 point to show that N varies in this direction. The constant N = 1 line, on which M varies, is drawn with the first segment highlighted in red and a character "M" drawn at the M = 2 point to show that M varies in this direction.

The associated text command is:

```
view highlight {on | off}
```

4.12 Blanking Mode

Selection of the "Blanking Mode" submenu causes the following pulldown menu to be displayed.



The setting of the "Blanking Mode" determines how the graphics driver interprets the existence of IBLANK data in the grid dataset. By default, "Blanking On - Show Grid" is set, so that points flagged as holes are not displayed. The user also has the option of drawing with only hole points drawn using the "Blanking On - Show Hole" option, or drawing all points using the "Blanking Off" option.

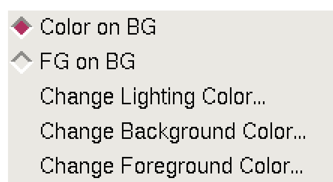
Releasing the left mouse button over the desired Blanking Mode will cause that radiobutton to be set, and the screen will be redrawn such that all currently displayed surfaces reflect their IBLANK values.

The associated text command is:

```
view blanking {off | grid | hole}
```

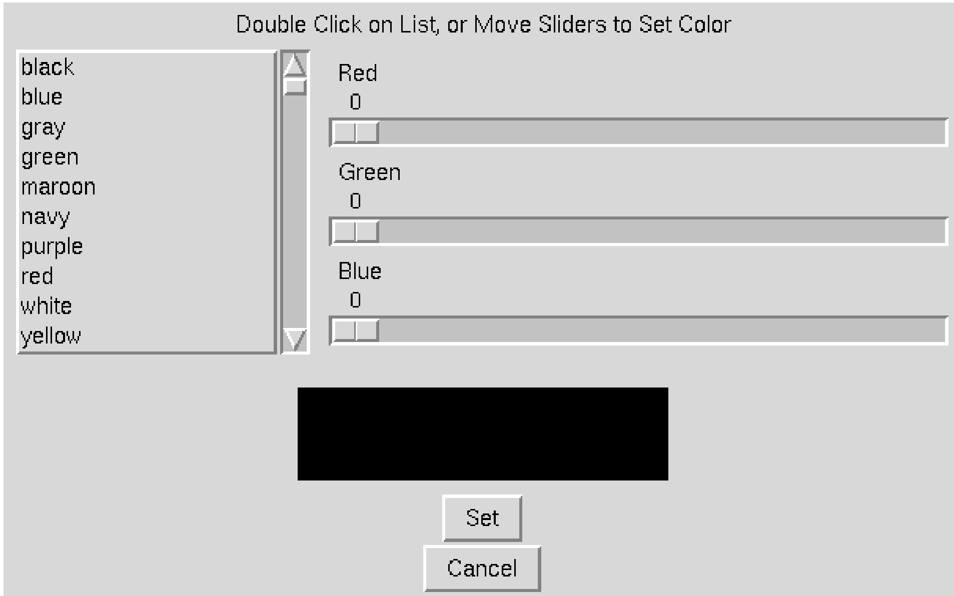
4.13 Color

Selection of the "Color" submenu causes the following pulldown menu to be displayed.



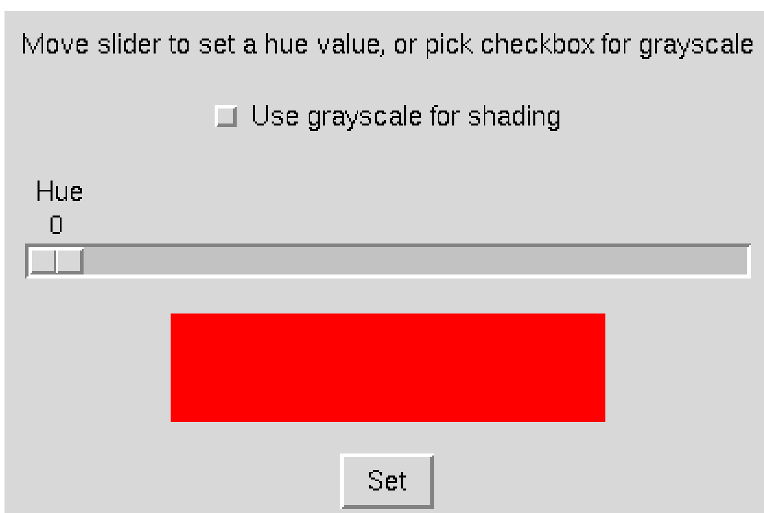
The MADCAP user has great flexibility in the choice of colors to be used for the foreground, background, or lighting models. The two radiobuttons at the top of the pulldown menu are used to cause the main graphics screen to be drawn either in full color on the current background color (Color on BG), or in the current foreground color on the current background color (FG on BG).

In full color mode, specific areas of the screen and the individual objects drawn are assigned colors by the program. To achieve a bi-color display such as black on white, the radiobutton should be set to "FG on BG", and the appropriate colors set using the "Change Background Color ..." and "Change Foreground Color ..." submenus at the bottom of this pulldown menu. Selecting either of these submenus causes the following window to be displayed.



In this window, the full list of X-graphics defined colors available on your system is shown in a scrollable list at the top left. To select a color from this list for either the foreground or background the user must double-click on the color name. This will cause the sliders to move to the corresponding Red, Green, and Blue (RGB) values making up that color. Alternatively, the user may use the sliders to modify the RGB values individually. The rectangle below the sliders reflects the currently set color. When the desired color has been found, select the "OK" button and the background or foreground color will be modified. Choose the "Cancel" button to escape with no action being taken. Using this window for both the foreground and background color, the user can achieve any bi-color display desired, such as red on yellow, orange on blue, or even DarkGoldenrod on PaleTurquoise.

The Color menu is also used to set the color to be used for solid bodies when the Global Display Mode (Section 4.6) or New Object Display Mode (Section 4.7) is set to "Lighted". When the "Change Lighting Color ..." option is chosen, the following window is displayed.



This window provides a checkbox to be used when the user desires a gray-scale for the lighting, as well as a slider to set a hue between 0 and 360. The hue value corresponds to a value on the color wheel, with both 0 and 360 corresponding to red. On this color wheel, yellow has a value of 60, green 120, cyan 180, blue 240, and magenta 300. Moving the slider causes the rectangle to be drawn in the color corresponding to the value set. If the grayscale checkbox is highlighted, the value on the Hue slider is not used.

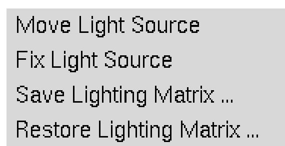
The associated text commands are:

`view color {coloronbg | fgonbg}`
`view fgcolor` *red green blue*
`view bgcolor` *red green blue*
`view lightcolor` *hue*

where *red*, *green*, and *blue* are values between 0. and 1., and *hue* is a value between 0. and 360., or -1 for grayscale.

4.14 Light Settings

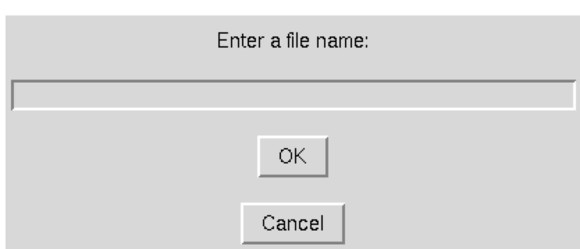
Selection of the "Light Settings" submenu causes the following pulldown menu to be displayed.



This menu is used to manipulate the light source location used when the Global Display Mode (Section 4.6) or New Object Display Mode (Section 4.7) is set to "Lighted". When the "Move Light Source" option is selected, the dials, spaceball, and graphics window rotation and translation buttons (see Section 2.2) cause the light source to move, rather than causing the geometry to move. The light source moves with the same types of body or screen-based transformations that the geometry uses. The geometry image is updated to reflect the new lighting in real time as the light source is moved. When large amounts of graphics data are displayed, this can be a slow process. It is recommended that the light source be set with the minimal amount of data displayed that is required to achieve the desired lighting.

To fix the light source at a particular location, and return the dials to manipulating the geometry, the "Fix Light Source" menu option should be chosen. A window will appear informing the user that the dial function has been set back to geometry manipulation.

Lighting matrices may be saved to file and restored from file with the last two menu options. Selecting either of these causes the following window to be displayed.



Entering a file name and choosing the “OK” button causes the current lighting matrix to be written to the specified file in “Save” mode, or read and loaded to the program as the current lighting matrix in “Restore” mode.

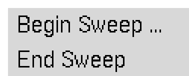
Lighting matrices may be manipulated at any time, regardless of whether the Global Display Mode (Section 4.6) or New Object Display Mode (Section 4.7) is set to “Lighted”. Of course, no affect is seen graphically unless Lighted mode is set.

The associated text command is:

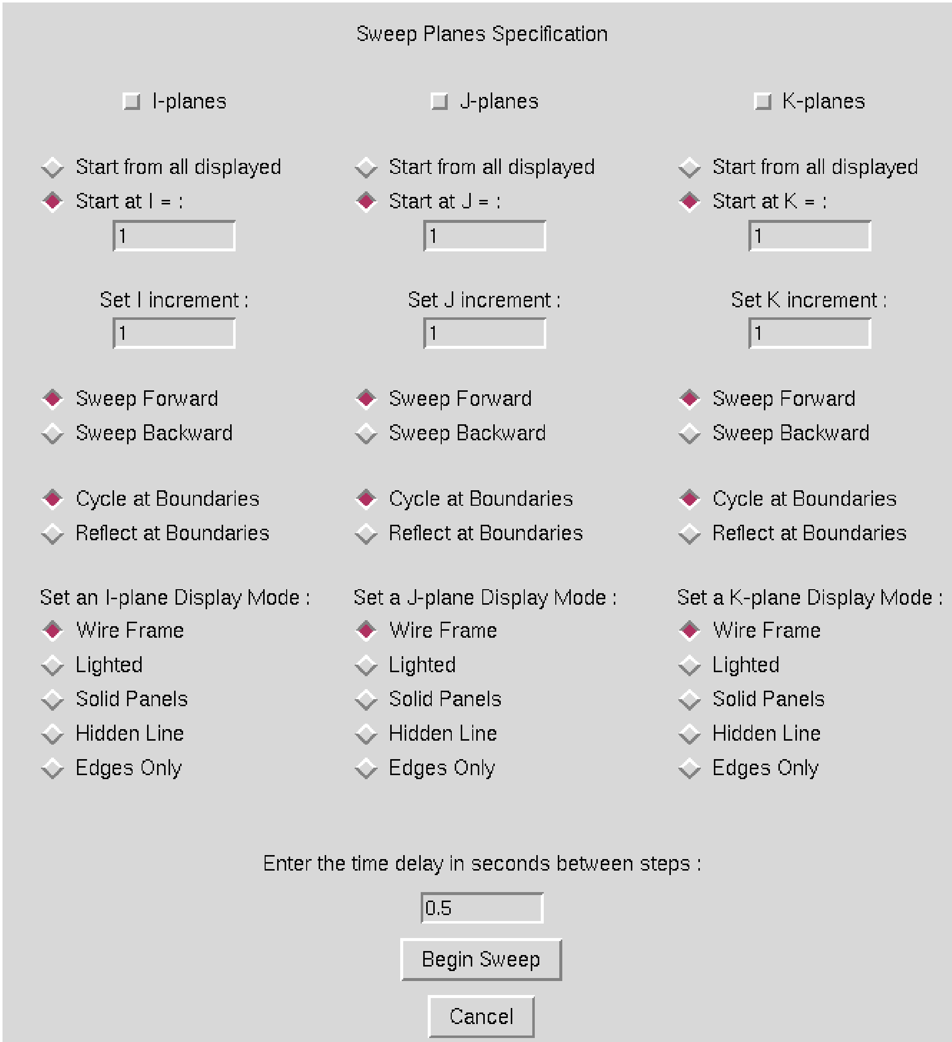
`view lightsource {move | fix | save` *filename* `| restore` *filename*`}`

4.15 Sweep Plane Settings

Selection of the “Sweep Plane Settings” submenu causes the following pulldown menu to be displayed.



This menu is used to access the automatic grid plane sweeping of structured zonal grids capability in MADCAP. Grid planes in the current display file can be made to automatically sweep, alternately turning on and off the planes requested. Selecting the “Begin Sweep ...” option causes the following window to be displayed.



Sweeps are performed on any combination of I, J, or K planes simultaneously, as indicated by the checkboxes set across the top of the window. For any type of plane, there are a number of additional options which may be set. First, a radiobutton must be set indicating whether the sweeps should originate from all displayed planes of the given type, or only from a specfic plane whose index is entered in the entry box. Next, the step increment for the sweep must be entered, and is defaulted to 1. For each plane type, the user may specify that the sweep progress forward or backward. Also the behavior at the 1 and max boundaries is specified with the "Cycle" or "Reflect" radiobuttons. "Cycle" will cause plane 1 to follow plane "max" in the sweep, while "Reflect" will cause plane "max − 1" to follow plane "max". A different display mode can be set for the planes in each direction, and is independent of the currently set Global Display Mode (Section 4.6) or New Object Display Mode (Section 4.7). Finally, a time delay between steps must be entered. It may be impossible to achieve very short time delays, depending on the amount of real time it takes to update the display on each step. If the specified elapsed time has not passed after the screen is updated, the program waits until it has passed. If the screen update has taken longer than the specified time, the program continues with the next update immediately.

When the desired sweep data has been entered, selecting the “OK” button will cause the sweep to begin. To exit the window with no action being taken, select the “Cancel” button.

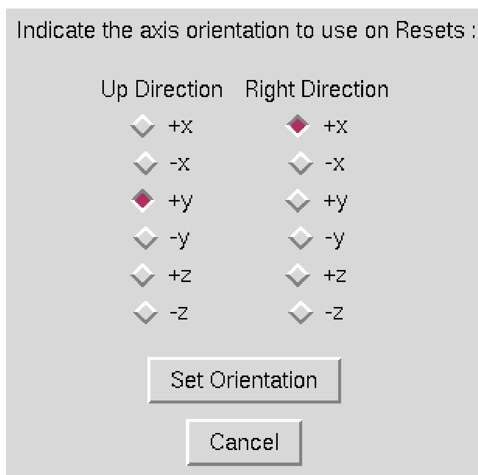
To stop the grid plane cycling, select the “End Sweep” option under the main GUI Sweep Planes menu. It may take an additional step before the cycling stops. Any grid planes displayed as a result of the cycling action remain displayed after the “End Sweep” has been selected.

The associated text command is:

```
view sweep time {ijk | ij | jk | ik | i | j | k} \
      {all | start_index} increment {for | back} \
      {cycle | reflect} {wireframe | edges | solid | hidden | lighted} \
     [{all | start_index} increment {for | back} \
      {cycle | reflect} {wireframe | edges | solid | hidden | lighted}] \
     [{all | start_index} increment {for | back} \
      {cycle | reflect} {wireframe | edges | solid | hidden | lighted}]
```

4.16 Axis Orientation on Reset ...

Selection of the “Axis Orientation on Reset” submenu causes the following window to be displayed.



MADCAP users have the option of defining the viewpoint to be used as the default axis orientation. This axis orientation is used whenever the user chooses to reset the view (Section 4.1), and is also the basis for any transformations resulting from setting view angles (Section 4.3). Thus, if the user prefers to always have a reset present a view from the front, with the up direction in the screen up direction, this can be set regardless of the coordinate system of the geometry by ensuring that the correct Up Direction and Right Direction are set in this window.

The default orientation is set by choosing a radiobutton from each column, Up Direction and Right Direction. The same axis cannot be set for both directions. The orientation of the third axis, into the screen or out of the screen, will be determined such that the right hand rule is followed. After the correct radiobuttons have been set, the “OK” button should be picked to set the default. This selection does not perform a reset, but only establishes a default orientation.

To escape from this window with no action being taken, select the “Cancel” button.

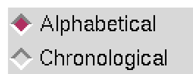
The associated text command is:

```
view axisorient {+x | -x | +y | -y | +z | -z } { +x | -x | +y | -y | +z | -z}
```

where the first direction specifies the up direction and the second specifies the right direction.

4.17 Surface File Sort Mode

Selecting the "Surface File Sort Mode" submenu causes the following pulldown menu to be displayed.



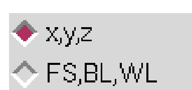
The geometry created in the surface geometry module of MADCAP is stored by an assigned name in the surface geometry file. At times it is desirable to access these names in the graphics window name list (Section 2.1) either alphabetically or chronologically. The "Surface File Sort Mode" option is used to determine how the surface names are presented to the user. In "Alphabetical" mode, newly assigned names are merged into an alphabetical presentation of the surface name list. In "Chronological" mode, newly assigned names are always appended to the bottom of the list. This mode also provides the user with a sense of how recently the surfaces have been created. The mode can be switched as desired at any time.

The associated text command is:

```
view sortmode {alpha | chron}
```

4.18 Axis Mode

Selection of the "Axis Mode" submenu causes the following pulldown menu to be displayed.



Choosing between the two radiobuttons switches the graphics display between plotting the pure x, y, z coordinate data existing in the open files to plotting the data in a fuselage station (FS), buttline (BL), waterline (WL) coordinate system. Relationships between x, y, z and FS,BL,WL coordinates are established under the File Set FS, BL, WL Directions menu option for each open file in the system, as described in Section 3.12.

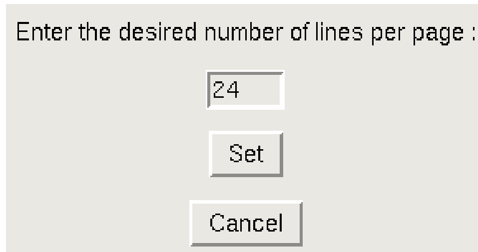
When the mouse is released over the requested coordinate system, the graphics are updated to reflect the selected option. In this manner, two files containing the same geometry but in different coordinate systems (for instance, y-up and z-up) can be viewed consistently.

The associated text command is:

```
view axismode {xyz | fsblwl}
```

4.19 Set Output Lines Per Page

Some MADCAP operations raise the text window and scroll information to it. The user controls how many lines are output before being prompted to continue or end with this view command. The window shown below is presented after selecting the command from the View menu.



Simply enter the number of lines per page desired, and hit the OK button. This will set the lines per page for any scrolling output function desired in MADCAP. To escape with no action being taken, pick the Cancel button.

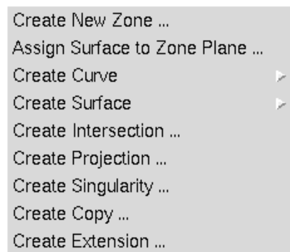
The associated text command is:

`view lines_per_page` *nlines*

5 Create Menu

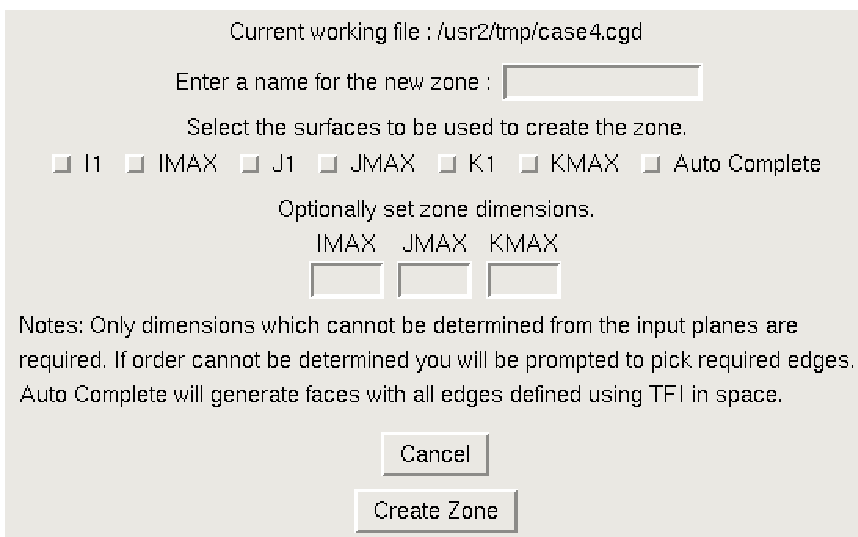
The Modular Aerodynamic Design Computational Analysis Process (MADCAP) "Create" library and menus are used to create new objects such as points, curves, surfaces, and zones. Typically, objects created using this menu are made "from scratch", whereas objects created by using the Manipulate Menu (see Section 6) are composed of portions of the object from which they are derived.

Selection of the "Create" menu item produces the drop down menu shown below.



5.1 Create New Zone

Selection of the "Create New Zone ..." submenu causes the following window to be displayed.



This operation simply creates a new zone and defines its name and dimensions. If the working file is a *.cgd* file, the zone will be created of the specified size, and all points in the zone will be at (0.,0.,0.). More typically, this operation is used when the working file is a *.csf* file, to define a new, empty zone whose faces will be assigned as they are created using other program functions. (See Section 5.2.)

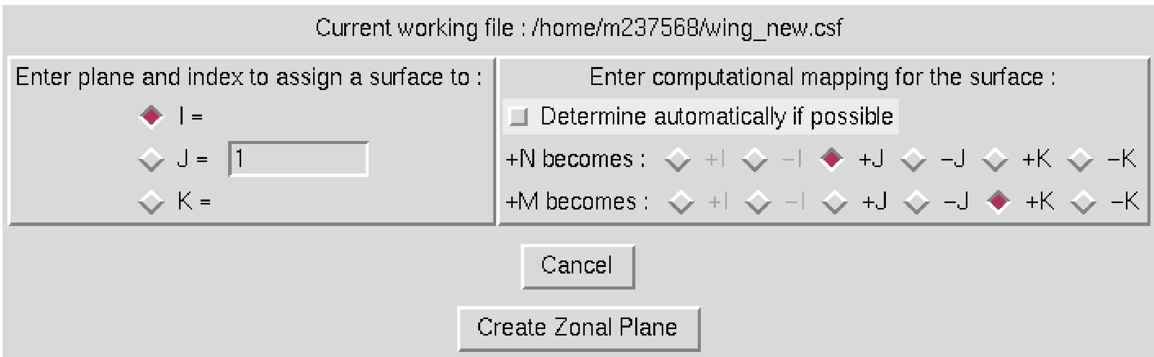
To create a new zone definition, the user simply needs to enter a name and dimensions for each of the three computational directions in the appropriate entry boxes, and then click the "Create Zone" button. To escape from this window with no action being taken, select the "Cancel" button.

The text commands generated by this operation are :

```
create zone imax imax jmax jmax kmax kmax
file save name zonename
```

5.2 Assign Surface to Zone Plane

Selection of the "Assign Surface to Zone Plane ..." submenu causes the following window to be displayed.



This operation is used to assign a surface generated in MADCAP a definition in an existing zone. For example, through several other MADCAP create, manipulate (Section 6), or grid generation (Section 7) operations, the user may have generated a surface to serve as the J = 1 boundary of an existing zone. (If no zone exists for the boundary yet, it must be generated first using the Create Zone operation first, as described in Section 5.1). To make the assignment, this operation is used. The user would set the radiobutton to the left of the window to J, and ensure that a 1 is entered in the entry box for the J index being assigned. The user must also specify a mapping to transform the N, M computational indices of the surface to the appropriate I, J, and K computational indices of the zone. The Highlight Object Origins option under the view menu (see Section 4.11) is often helpful in making this determination. Once a single zone face exists in a zone, it is often possible for MADCAP to automatically determine the correct transformation for adjacent faces by comparing edges of the surfaces. If the user would like the program to determine the transformation, simply select the checkbutton on the right.

Selecting the "Create Zone Plane" button will cause the main program to prompt first for a surface to assign to a zone, and then for the zone the surface is to be assigned to. MADCAP will then create a new zone plane in the specified zone. If any edge mismatches occur where the new zone plane abuts existing zone planes, an error message is generated indicating the magnitude of the mismatch relative to local grid spacings. Small mismatches can usually be ignored, but larger mismatches should usually be investigated to determine the source of the problem. In any event, the zone plane is created even if mismatches exist along edges.

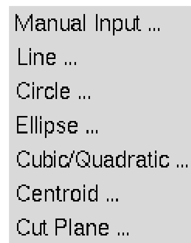
To escape from this window with no action being taken, select the "Cancel" button.

The associated text command is :

```
create zplane {i | j | k} index nmap {auto | +i | -i | +j | -j | +k | -k } \
   [mmap {+i | -i | +j | -j | +k | -k}]
```

5.3 Create Curve

Selection of the “Create Curve” submenu causes the cascade menu shown below to be displayed.

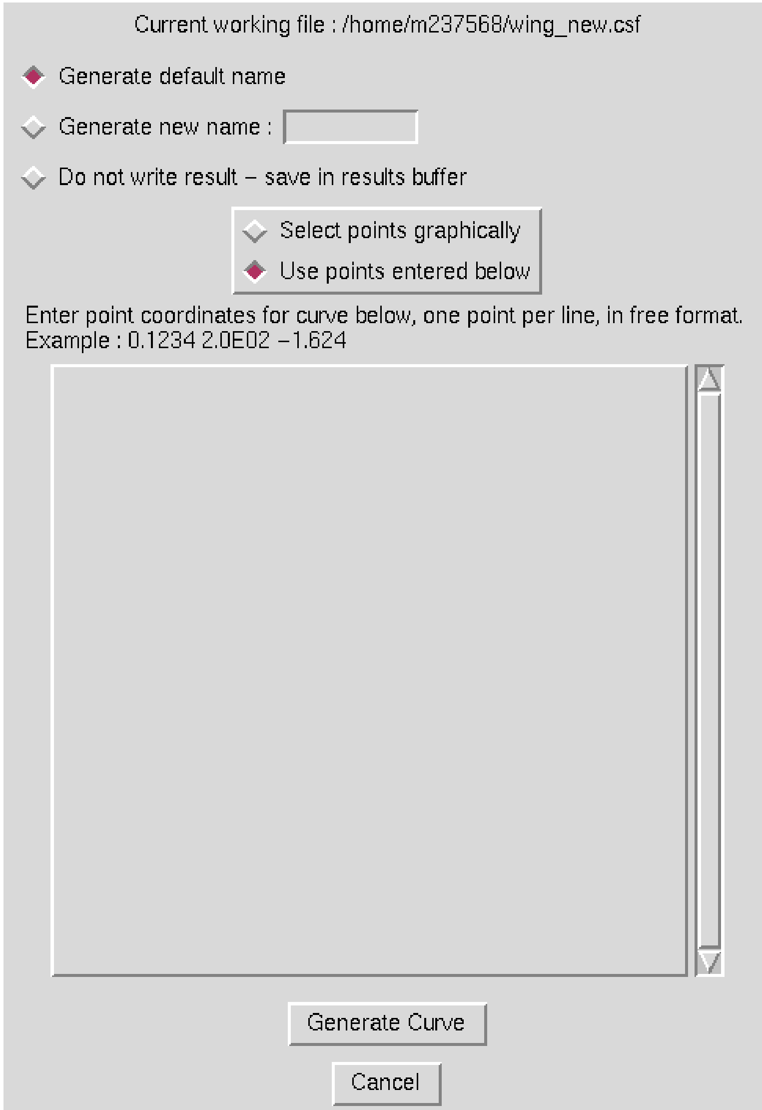


Seven different methods are provided for creating independent curves from scratch. These methods are:

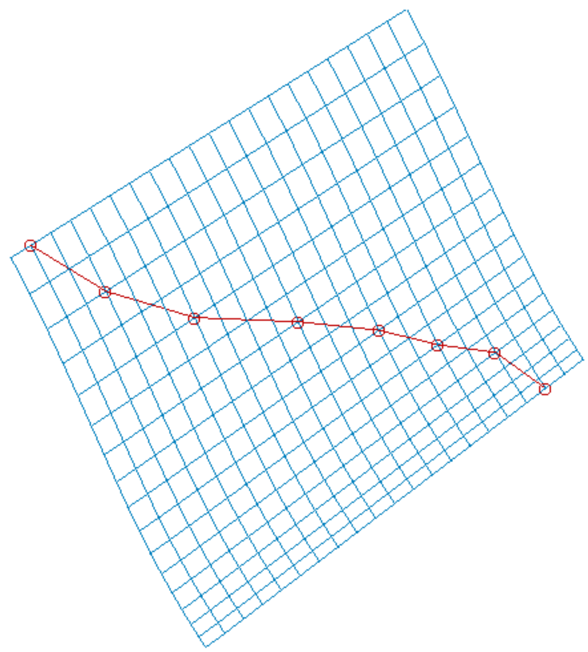
- Manual Input ...
- Line ...
- Circle ...
- Ellipse ...
- Cubic/Quadratic ...
- Centroid ...
- Cut Plane ...

5.3.1 Manual Input

Selection of the “Manual Input ...” submenu causes the following window to be displayed.



This operation is used to create a curve completely from scratch. The individual points making up the curve can either be input graphically from the screen or by typing in the coordinates desired in the large entry box in the window. The radiobutton must be set to match the style of input desired. If points are typed in at this window, they should be entered in free format, one point per line. Note also that the standard X-windows cut and paste utilities can be used to insert a multi-lined file in this format into the window. The user should also select the appropriate radiobutton at the top of the screen to have the program either generate a default name, use a specified new name, or simply save the result in the object manager results buffer. Finally, the user should select the "Generate Curve" button. If the points are to be entered graphically, the program will prompt for points to make up the curve until the "done" button on the main graphics screen is selected. The red curve in the image below is an example of a manual-input curve defined by graphically selecting specific points from the larger surface to generate a new curve.

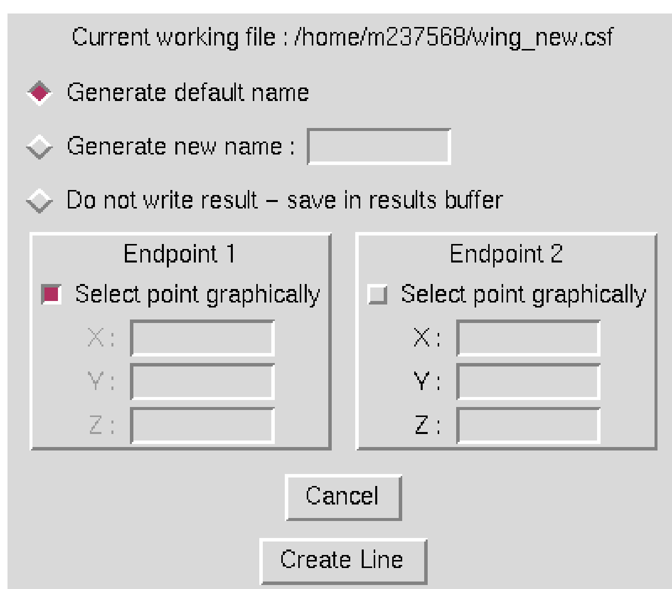


The text commands generated are :

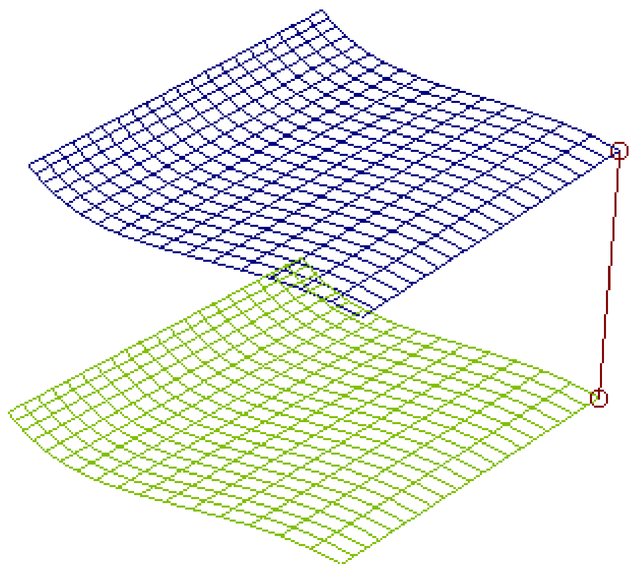
`om select point` *point_definition* (Multiple instances)
`create manual_curve`
`file save {default | name` *curvename*`}`

5.3.2 Line

Selection of the “Line ...” submenu causes the following window to be displayed.



This operation is used to create a curve consisting of simply a straight line between two endpoints. The endpoints may either be selected graphically by setting the checkbuttons in the window, or the x, y, and z values for the endpoints may be set in the entry boxes. Radiobuttons should also be set to generate either a default name, a specified name, or to save the line in the object manager results buffer. When the "Create Line" button is selected, the program creates the line or prompts for inputs first if the endpoints are being selected graphically. The red line in the image below is an example of a line generated by graphically selecting the corner points of two displayed surfaces.

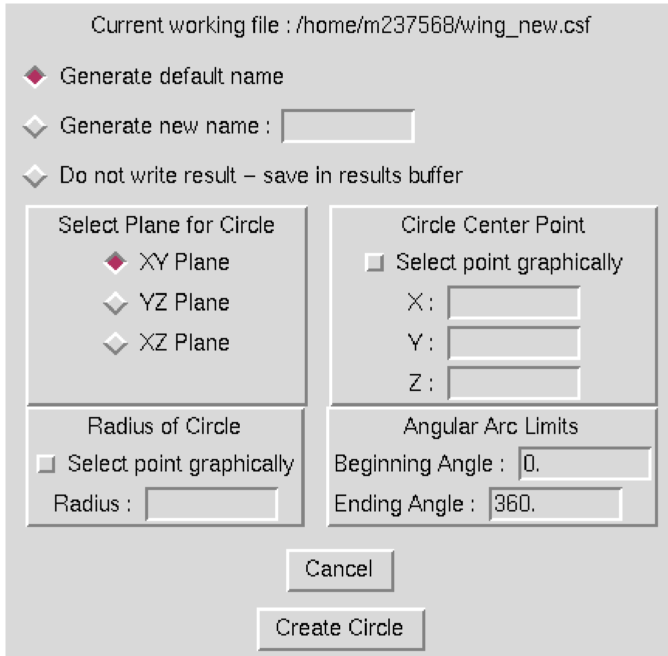


The text commands generated are :

`om select point` *point_definition* (Two instances)
`create line curve`
`file save {default | name` *curvename*`}`

5.3.3 Circle

Selection of the "Circle ..." submenu causes the following window to be displayed.



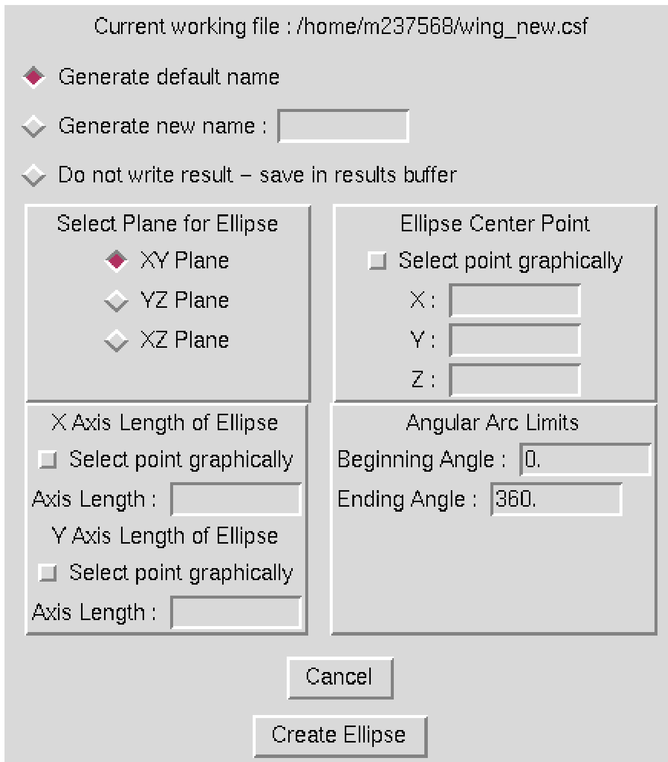
This operation is used to create a curve consisting of a circular arc. Circular arcs can be generated parallel to the xy plane, yz plane, or xz plane by selecting the appropriate radiobutton. A center for the circle must be defined either by entering coordinates in the entry boxes or having the program prompt to select the center from the graphical display. Similarly, the radius may be entered in the entry box or a point may be picked from the graphics screen which will define the radius relative to the centerpoint. Finally, the portion of the defined circle to be generated is defined by setting the angular arc limits between 0 and 360 degrees. Radiobuttons should also be set to generate either a default name, a specified name, or to save the line in the object manager results buffer. When the "Create Circle" button is selected, the program will either prompt for graphical selections or generate the defined circular arc. A point is generated every 1 degree of the circular arc. The red curve in the image below is an example of a circular arc defined with the center at the midpoint of the first end of the two lines and a radius defined by selecting the first endpoint of one of the lines. The beginning and ending arc values are defined in a standard right-handed sense. For the circle below in the yz plane, the angle limits entered were 90 and 270 degrees.

The text commands generated are :

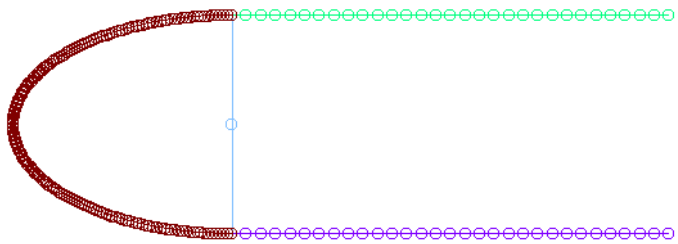
```
om select point point_definition (One instance, to define center)
om select point point_definition (One instance, if defining radius graphically)
create circle plane {xy | xz | yz} [radius radius] start angle end angle
file save {default | name curvename}
```

5.3.4 Ellipse

Selection of the "Ellipse ..." submenu causes the following window to be displayed.



This operation is used to create a curve consisting of an elliptical arc. Elliptical arcs can be generated parallel to the xy plane, yz plane, or xz plane by selecting the appropriate radiobutton. A center for the ellipse must be defined either by entering coordinates in the entry boxes or having the program prompt to select the center from the graphical display. Similarly, the two axis lengths may be entered in the entry box or points may be picked from the graphics screen which will define the axes relative to the centerpoint. Finally, the portion of the defined ellipse to be generated is defined by setting the angular arc limits between 0 and 360 degrees. Radiobuttons should also be set to generate either a default name, a specified name, or to save the line in the object manager results buffer. When the "Create Ellipse" button is selected, the program will either prompt for graphical selections or generate the defined elliptical arc. A point will be generated every 1 degree on the elliptical arc. The red curve in the image below is an example of an elliptical arc defined with the center at the midpoint of the first end of the two lines, one axis defined by selecting the first endpoint of one of the lines, and the other axis defined by a keyed in length. The beginning and ending arc values are defined in a standard right-handed sense. For the ellipse below in the yz plane, the angle limits entered were 90 and 270 degrees.

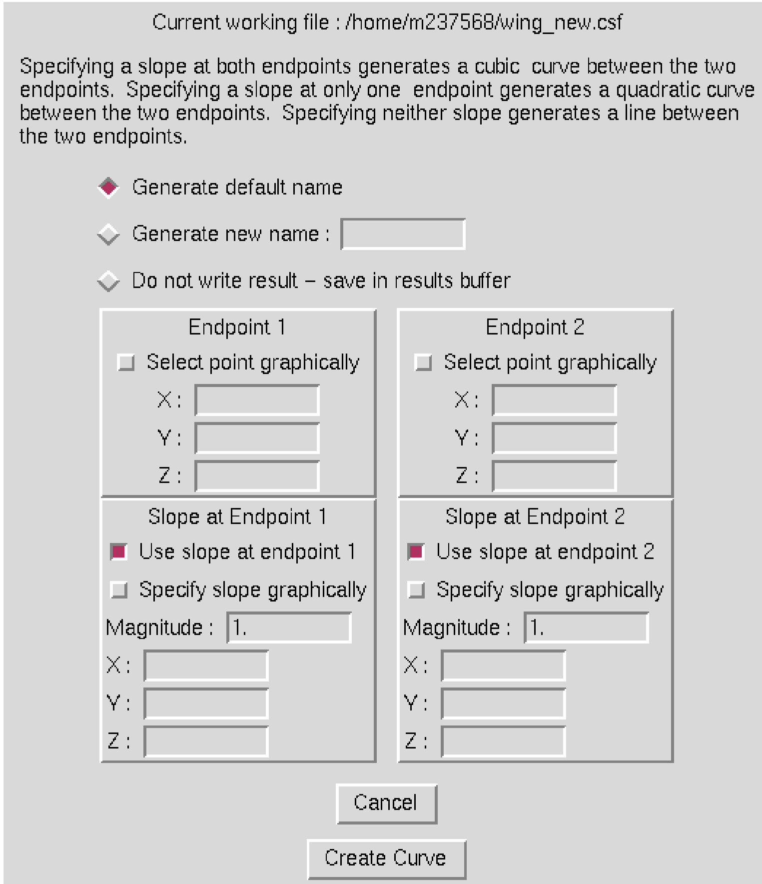


The text commands generated are :

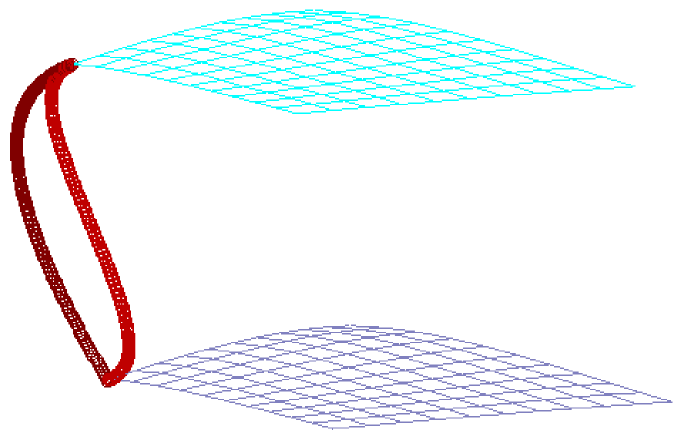
`om select point` *point_definition* (One instance, to define center)
`om select point` *point_definition* (Additional instances, if defining axes graphically)
`create ellipse plane {xy | xz | yz} [aaxis` *aaxis*`] [baxis` *baxis*`] \`
`start` *angle* `end` *angle*
`file save {default | name` *curvename*`}`

5.3.5 Cubic/Quadratic

Selection of the "Cubic/Quadratic ..." submenu causes the following window to be displayed.



This operation is used to create either cubic or quadratic curves. The user must define the two endpoints for the curve, either by selecting the checkbutton to select them graphically or by entering coordinates in the entry boxes. To define a cubic curve, the checkbuttons to use the slope information at *both* endponts should be selected. For a quadratic curve, slope information is included at only one of the endpoints. Slopes may also be defined either graphically or by entering the components of the slope vector relative to (0,0,0) in the entry boxes. If slopes are defined graphically, the user is prompted for two points which define the slope vector direction. The magnitude entry boxes must be filled out for the ends where slope is being defined. This value controls the shape of the curve. Higher values cause the slope direction to be maintained for longer arc length of the final curve than lower values. Radiobuttons should also be set to generate either a default name, a specified name, or to save the line in the object manager results buffer. When the "Create Curve" button is selected, the program will either prompt for graphical selections or generate the defined cubic or quadratic. The red curves in the image below are examples of cubic and quadratic curves generated between two surface points. For each curve, the slope was controlled at the top to match the slope of the surface, but the slope at the bottom was only controlled on the cubic curve. Note that if slope is controlled at neither end, the operation defaults to generating a line between the two endpoints, just as if the Create Line menu option (Section 5.3.2) had been selected.



The text commands generated are :

```
om select point point_definition (Two instances, to define endpoints)
om select point point_definition (Two instances per slope, if defining slopes graphically)
```

then

```
create cubic curve [slope1 x y z] [slope2 x y z] first_mag magnitude \
   last_mag magnitude
```

or

```
create quadratic curve slope_end {first | last} [slope x y z] \
   magnitude magnitude
```

or

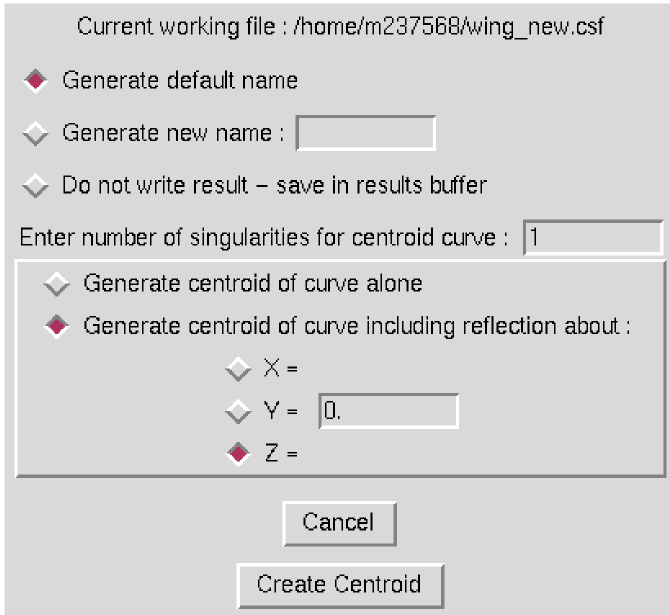
```
create line curve
```

and finally

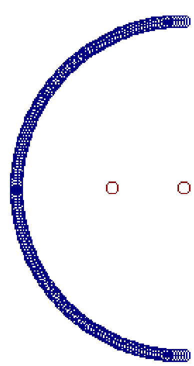
```
file save {default | name curvename}
```

5.3.6 Centroid

Selection of the “Centroid ...” submenu causes the following window to be displayed.



This operation is used to generate the centroid of a curve. Optionally, the user can generate a singularity at the centroid by specifying how many times to duplicate the point generated as the centroid. Also, the user may generate the true centroid of the curve, or the centroid of the curve including its reflection about any plane perpendicular to a principal plane. This is especially useful when working with only one half of a symmetric geometry. In this case, the radiobutton for which type of symmetry plane is being used must be set, as well as a value for the location of the plane. Radiobuttons should also be set to generate either a default name, a specified name, or to save the centroid in the object manager results buffer. When the "Create Centroid" button is hit, the program will prompt for the curve to generate a centroid for. The two red points in the image below show the results of generating a centroid for the circular arc with and without using the reflection option.

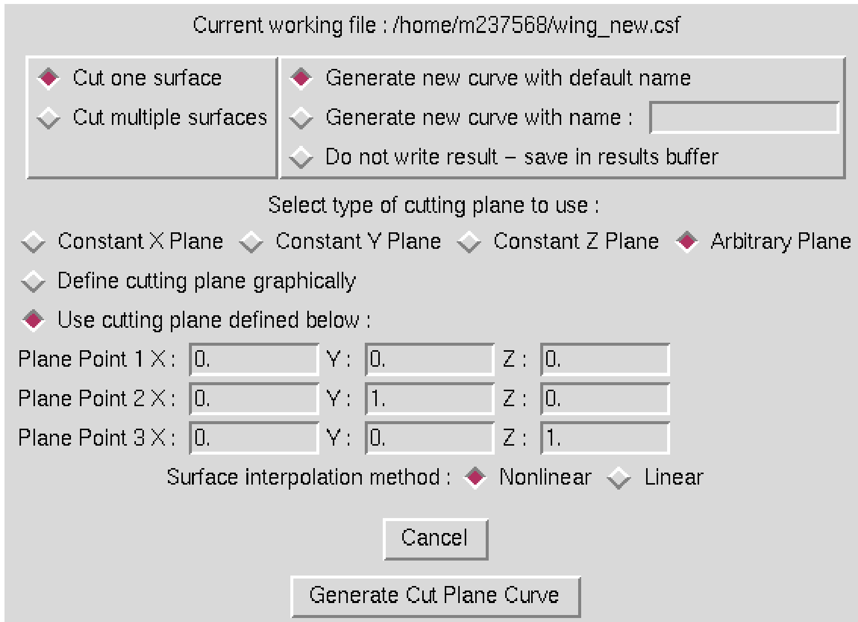


The text commands generated are :

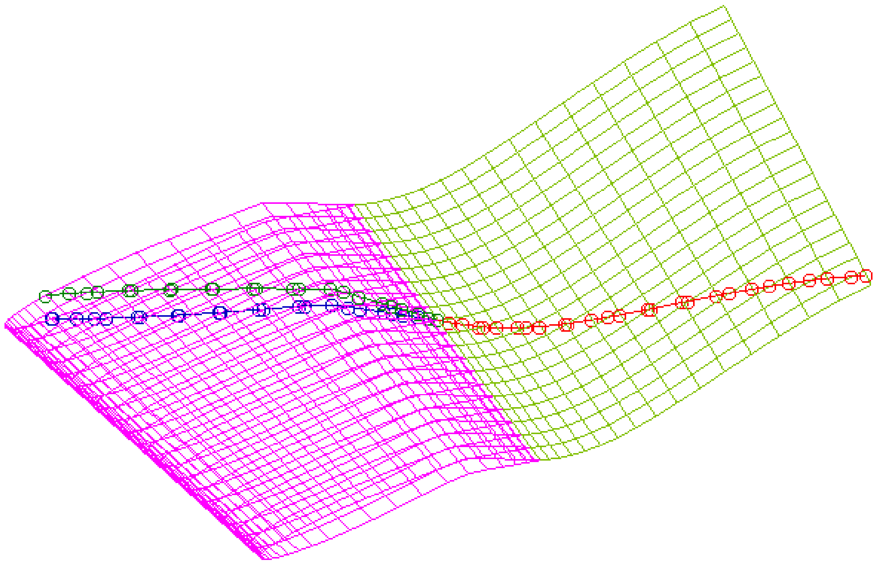
om select curve *curve_definition*
create centroid curve nsing *n_singularities* [refplane {x | y | z} *value*]
file save {default | name *curvename*}

5.3.7 Cut Plane

Selection of the “Cut Plane ...” submenu causes the following window to be displayed.



This operation is used to generate a curve by passing a cutting plane through a surface or set of surfaces. The user should first indicate whether to cut a single surface or multiple surfaces using the radiobuttons at the top left of the window. The type of plane to be used for cutting should then be set to either a constant x, y, or z plane, or an arbitrary plane. Only a single point is required to define the cutting plane for the first three options, while three points are required to define an arbitrary plane. The user may choose to define the point or points defining the cutting plane graphically or by entering coordinates in the entry boxes shown. The user must also specify whether to use linear or nonlinear interpolation when cutting through the cells defining the surface being cut. Each surface cut generates a separate curve, and if a cut through a surface generates multiple curves, each curve is output individually. Radiobuttons should also be set to generate either a default name, a specified name, or to save the centroid in the object manager results buffer. When the “Create Cut Plane Curve” button is selected, the program prompts for the surfaces to be cut, and the point(s) defining the plane if they are to be entered graphically. The red and blue curves in the image below show an example of a cutting plane through multiple surfaces.

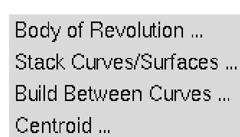


The text commands generated are :

`om select surface` *surface_definition* (Multiple instances, to define surfaces being cut)
`om select point` *point_definition* (One or three instances, to define cutting plane)
`create cutplane {x | y | z | arb} interp {linear | nonlinear}`
`file save {default | name` *curvename*`}`

5.4 Create Surface

Selection of the "Create Surface" submenu causes the cascade menu shown below to be displayed.

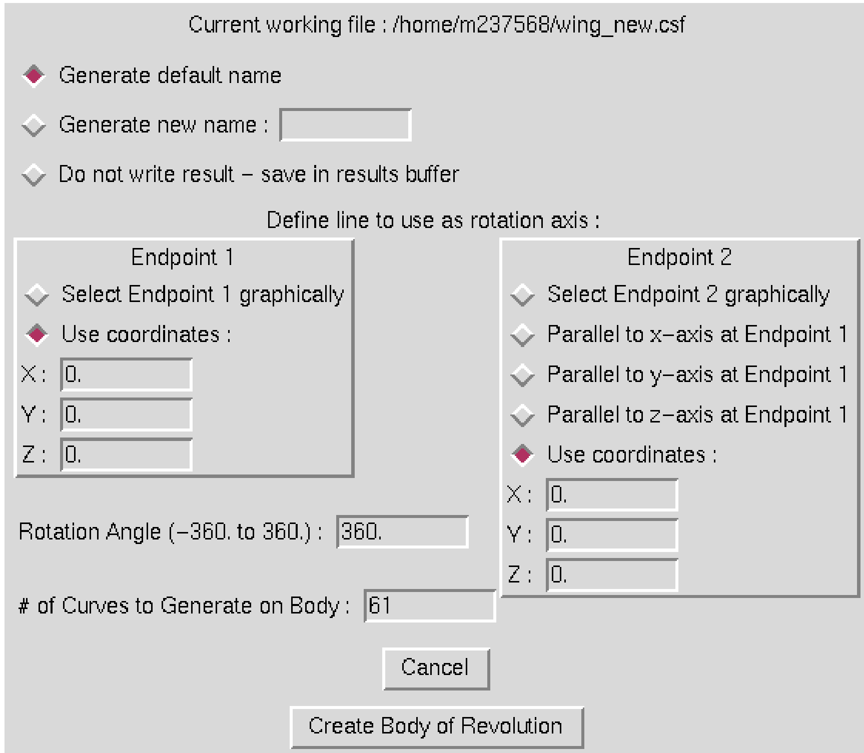


Four different methods are provided for creating surfaces entirely from scratch. These methods are:

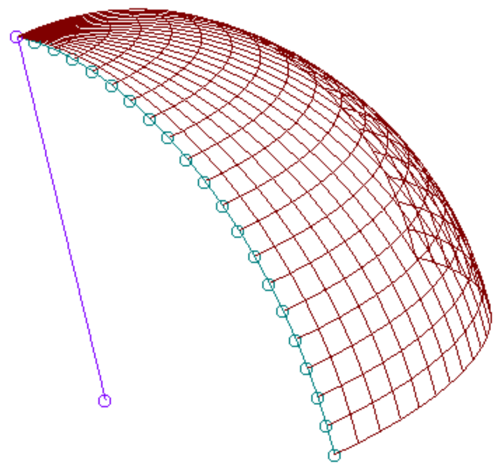
- Body of Revolution ...
- Stack Curves/Surfaces ...
- Build Between Curves ...
- Centroid ...

5.4.1 Body of Revolution

Selection of the "Body of Revolution ..." submenu causes the following window to be displayed.



This operation is used to generate a surface by rotating a curve about a line an input angular distance and saving the surface thus swept out. The user must set the radiobutton for the first endpoint of the line to rotate about to either identify the endpoint graphically or use the coordinates entered into the entry boxes. To define the rotation line, the user can then specify that the line is simply parallel to either the x, y, or z axes at the first endpoint, or may again select an endpoint graphically or by using the entry boxes. The rotation angle (0 to 360 degrees) must then be entered, as well as the number of curves to generate on the resulting surface. The curve being rotated will become the $N = 1$ line on the surface, and N_{max} will be the number entered here. (M_{max} will remain the number of points on the curve being rotated.) Radiobuttons should also be set to generate either a default name, a specified name, or to save the body of revolution in the object manager results buffer. When the "Create Body of Revolution" button is selected, the program prompts for the curve to be rotated, and the point(s) defining the rotation line if they are to be entered graphically. The red surface in the image below shows an example of a body of revolution generated by rotating the blue curve 90 degrees about the purple line.

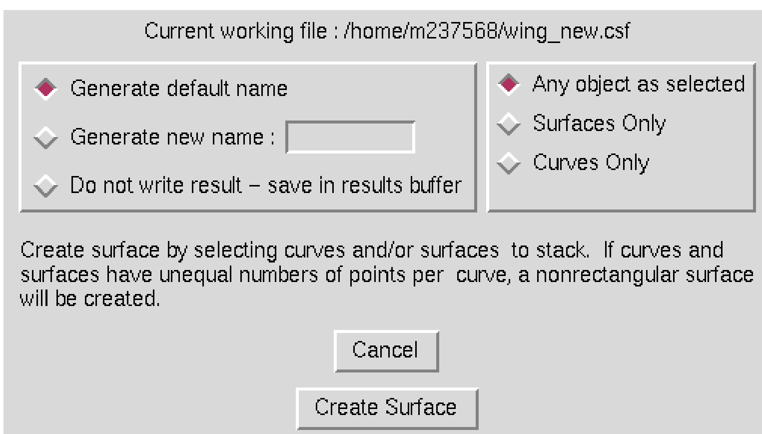


The text commands generated are :

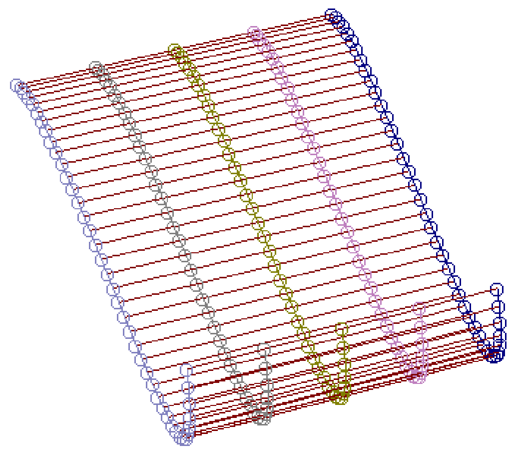
`om select curve` *curve_definition* (One instance, to define curve to rotate)
`om select point` *point_definition* (One or two instances, to define line to rotate about)
`create body_of_rev parallel_to {line | x | y | z} angle` *value* `ncrvs` *value*
`file save {default | name` *surfacename*`}`

5.4.2 Stack Curves/Surfaces

Selection of the "Stack Curves/Surfaces ..." submenu causes the following window to be displayed.



This operation is used to generate a surface by simply stacking several independent curves as constant N-lines to create a new surface. Thus, the order of points on the curves selected as well as the order in which these curves are selected are both critical in determining the definitions of the new surface generated. Since surfaces can be viewed as simply a collection of previously stacked curves, it is even possible to include surfaces in the select buffer. In this case, the individual N-lines from the surface are stacked in the new surface just as if they had each been selected individually. The user needs only to specify whether the program should interpret picks on surfaces as selecting the specific curve picked on or as selecting the entire surface for inclusion in the final surface. The user should also set the radiobuttons at the upper left to define whether to generate a default name, a specified name, or to save the new surface in the object manager results buffer. When the "Create Surface" button is selected, the program prompts for the curves and or surfaces to be stacked to generate the new surface. The red surface in the image below shows an example of a surface generated by stacking the individual displayed curves. Note that in this image, the N-lines on the resulting surface are obscured by overplotting the individual curves stacked to create the surface to show how the operation works. Without these curves displayed, the N-lines on the resultant surface would also be drawn in red.



The text commands generated are :

`om select curve` *curve_definition* (Multiple instances)

and/or

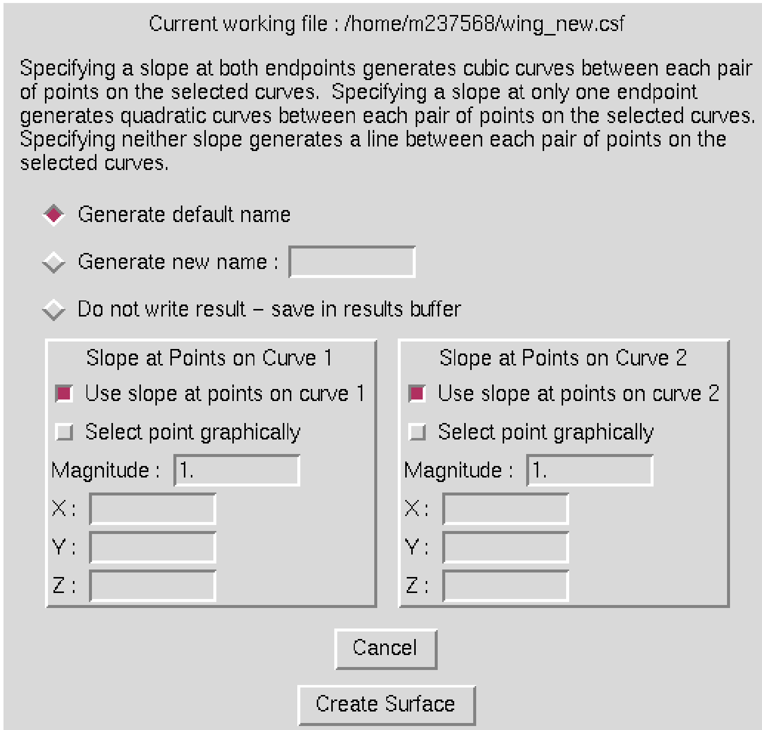
`om select surface` *surface_definition* (Multiple instances)

followed by

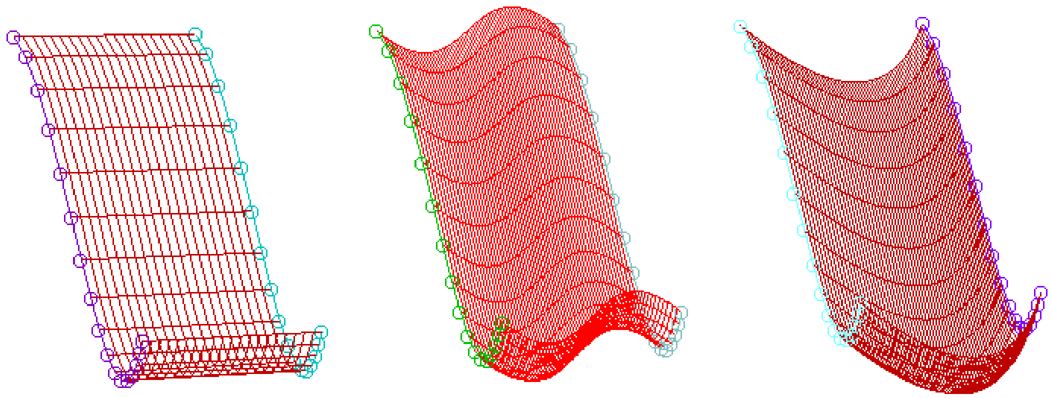
`create stack`
`file save {default | name` *surfacename*`}`

5.4.3 Build Between Curves

Selection of the "Build Between Curves ..." submenu causes the following window to be displayed.



This operation is used to generate a surface between two curves containing the same number of points. The surface generated can consist of straight lines, cubic curves, or quadratic curves between pairs of curve points. To generate straight lines, set the checkbuttons to use the slope at neither of the two curves being built between. For a cubic, slopes must defined at both curves, and for a quadratic, at either one of the curves. Slopes can be defined either graphically or by entering the components of the slope vector relative to (0,0,0) in the entry boxes. If slopes are defined graphically, the user is prompted for two points which define the slope vector direction. The magnitude entry boxes must be filled out for the curves where slope is being defined. This value controls the shape of the final surface. Higher values cause the slope direction to be maintained for a longer arc length of the final surface than lower values. Radiobuttons should also be set to generate either a default name, a specified name, or to save the surface in the object manager results buffer. When the "Create Surface" button is selected, the user will be prompted for the two curves to be built between as well as any points to define slopes graphically. The red surfaces in the images below are examples of building linear, cubic, and quadratic surfaces between the end curves.



The text commands generated are :

> `om select curve` *curve_definition* (Two instances, to define curves to build between)
> `om select point` *point_definition* (Two instances per slope, if defining slopes graphically)

followed by

> `create cubic surface [slope1` *x y z*`] [slope2` *x y z*`] first_mag` *magnitude* `\`
> `last_mag` *magnitude*

or

> `create quadratic surface slope_end {first | last} [slope` *x y z*`] \`
> `magnitude` *magnitude*

or

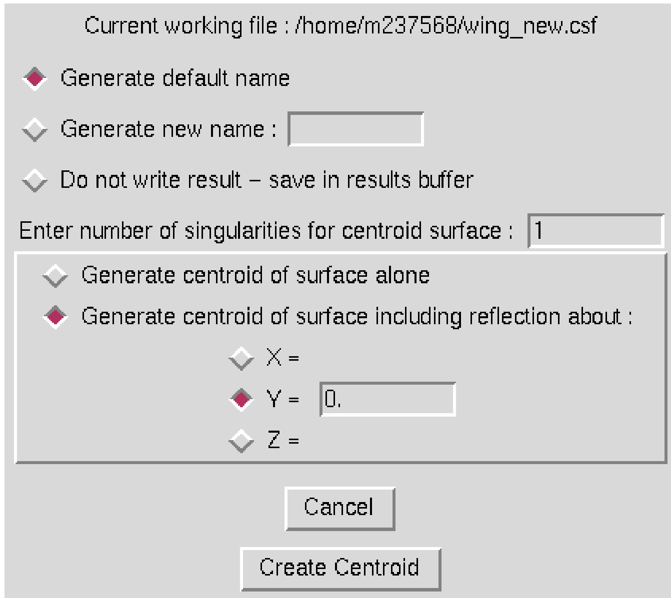
> `create line surface`

and finally

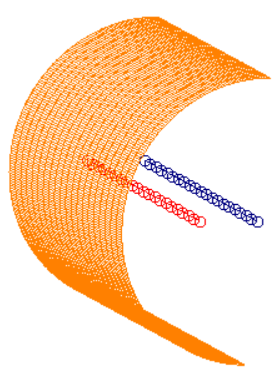
> `file save {default | name` *curvename*`}`

5.4.4 Centroid

Selection of the "Centroid ..." submenu causes the following window to be displayed.



This operation is used to generate a curve or a singular surface composed of the centroids of the constant N-lines on a surface. Optionally, the user can generate a singular surface by specifying how many times to duplicate the curve generated at the curve centroids. Also, the user may generate the true centroids of the N-lines, or the centroids of the N-lines including the reflection about any plane perpindicular to a principal plane. This is especially useful when working with only one half of a symmetric geometry. In this case, the radiobutton for which type of symmetry plane is being used must be set, as well as a value for the location of the plane. Radiobuttons should also be set to generate either a default name, a specified name, or to save the centroid in the object manager results buffer. When the "Create Centroid" button is hit, the program will prompt for the surface to generate a centroid for. The blue and red curves in the image below show the results of generating a centroid for the surface with and without using the reflection option.

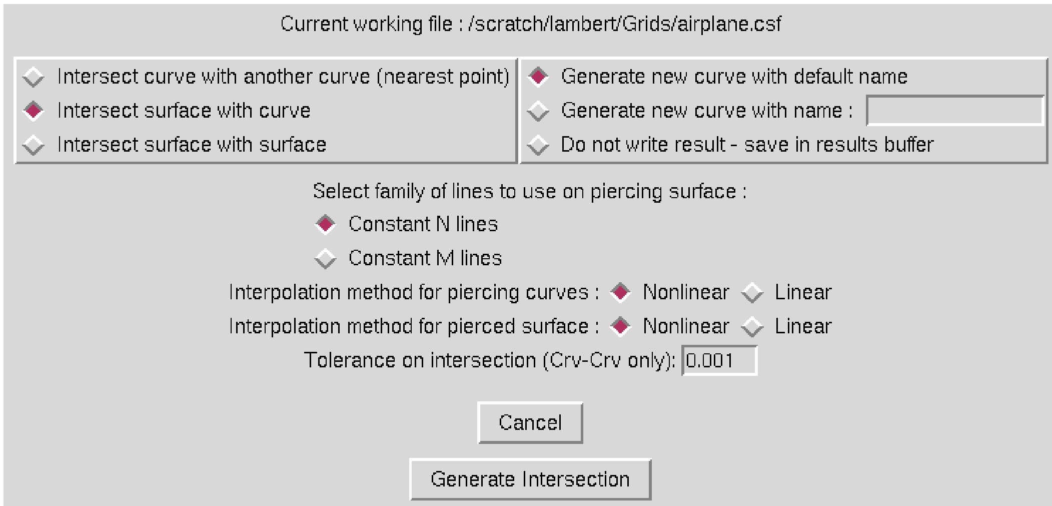


The text commands generated are :

`om select surface` *surface_definition*
`create centroid surface nsing` *n_singularities* `[refplane {x | y | z}` *value*`]`
`file save {default | name` *curvename*`}`

5.5 Create Intersection

Selection of the “Create Intersection ...” submenu causes the following window to be displayed.

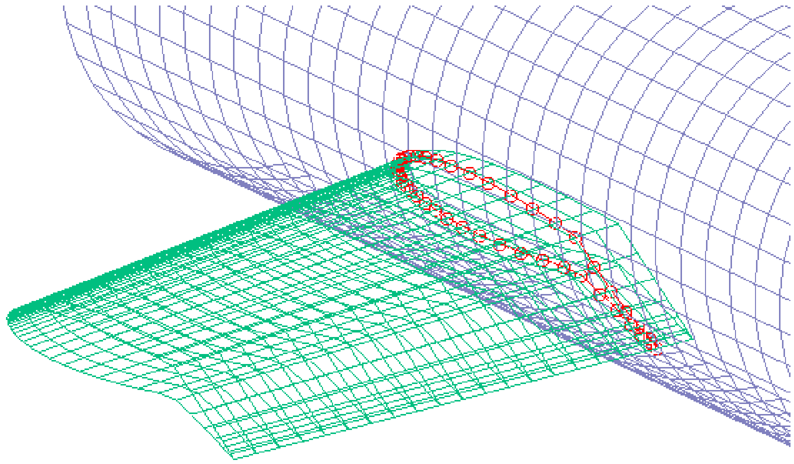


This operation is used to generate a curve by the intersection of two surfaces or the intersection of a surface with a family of curves on a surface, or to create a point by the intersection of two curves. The intersection mode is set by selecting the corresponding radio button at the top left of the window.

If the intersection of two curves is selected, the user may specify a tolerance for this intersection in the text field at the bottom of the window.

If surface-surface intersection is selected, the user must set which family of lines to use to create the intersection. Both these piercing curves and the intersected, or pierced, surface can be interpolated on using linear or nonlinear interpolation as set with the radiobuttons at the bottom of the window.

Note that for surface-surface intersections, it is possible to get four different definitions of the intersection curve, depending on which surface is used as the piercing surface and the family of grid lines used to pierce with. (Pierce with N-lines on surface 1, M-lines on surface 1, N-lines on surface 2, or M-lines on surface 2.) It is typically best to select whichever family of curves would result in the best definition of an intersection for the piercing curve. Radiobuttons should also be set to generate either a default name, a specified name, or to save the new curve in the object manager results buffer. When the “Create Intersection” button is hit, the program will prompt for the two objects to intersect. The red curve in the image below shows the result of intersecting the other two displayed surfaces.



The text commands generated are :

`om select surface` *surface_definition* (Two instances)

or

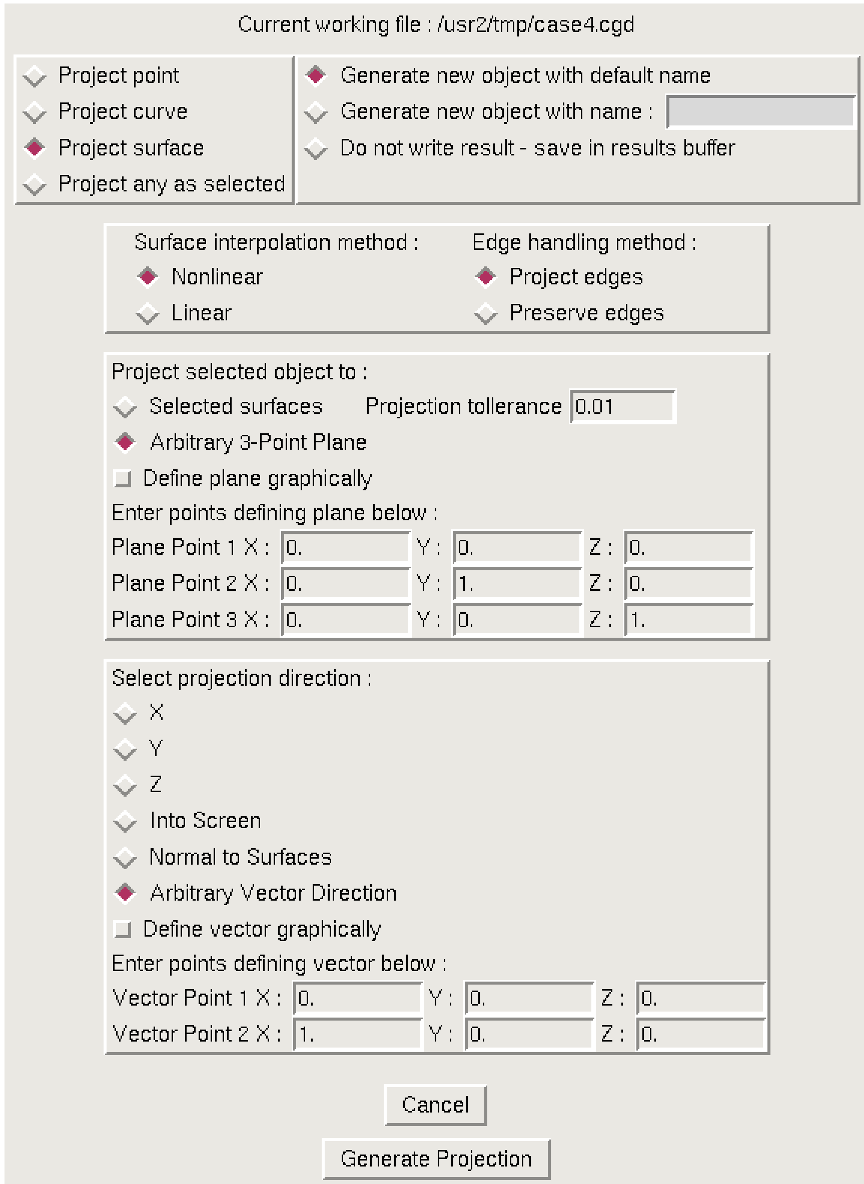
`om select curve` *curve_definition*
`om select surface` *surface_definition*

followed by

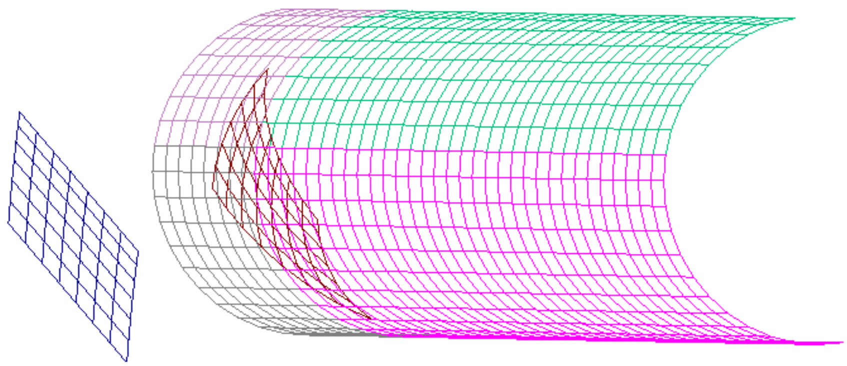
```
create intersection linetype {n | m} interp1 {linear | nonlinear} \
   interp2 {linear | nonlinear}
file save {default | name curvename}
```

5.6 Create Projection

Selection of the "Create Projection ..." menu causes the following window to be displayed.



This operation is used to generate the projection of an object to a plane or a set of existing surfaces. For projection to surfaces, the user must first define how to interpolate on the surfaces, linear or nonlinear. Since often it is desirable to project only the interior points of a surface and leave the edges untouched, radiobuttons are also provided to allow this. In the middle of the window, the user sets whether to project to a plane or a set of surfaces. If projecting to a plane, it may be defined graphically by picking three points on the display, or the coordinates may be entered in the entry boxes. The last area of the window controls the direction for the projection. The projection can be made in the x, y, or z directions or in an arbitrary direction. Arbitrary directions can be defined by specifying a direction into the current graphics screen, by selecting two points on the screen to define a vector, or by entering coordinates defining a vector direction into the window entry boxes. Projection can also be made normal to the surfaces projecting to (that is, to the closest point on the set of surfaces projecting to). Radiobutton should also be set to generate either a default name, a specified name, or to save the new curve in the object manager results buffer. When the "Create Projection" button is hit, the program will prompt for the object to project, as well as the points defining a plane and the vector direction if these options are set. The red surface in the image below shows the result of projecting the blue surface to the other displayed surfaces.

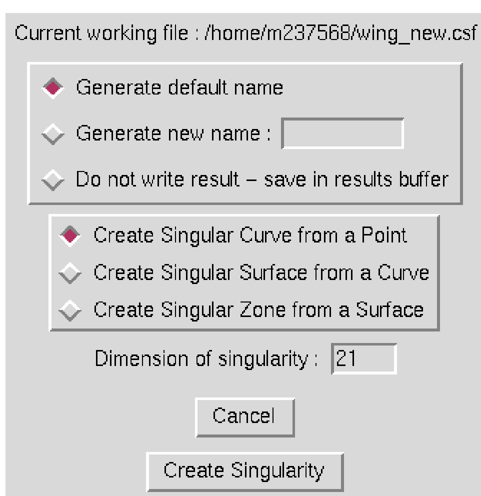


The text commands generated are :

om select *objecttype* *object_definition*
om select point *point_definition* (Multiple instances, to define projection plane)
om select point *point_definition* (Multiple instances, to define projection direction)
create project to {plane | surfaces} direction {x | y | z | normal | arb} \
  interp {linear | nonlinear} edges {projected | original}
file save {default | name *objectname*}

5.7 Create Singularity

Selection of the "Create Singularity ..." submenu causes the window below to be displayed.



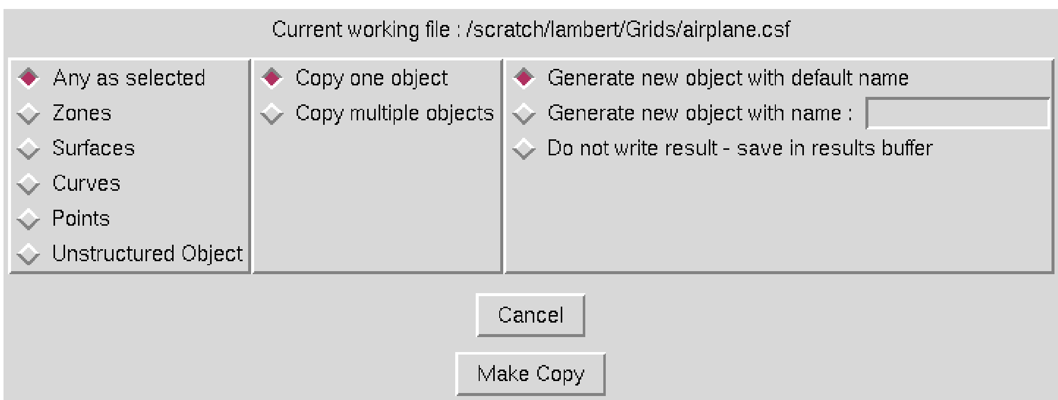
This operation is used to generate a singular object one order higher than the selecting object by duplicating its definition an input number of times and making that number the next higher dimension. Thus, a singular curve can be made by duplicating a point, a singular surface can be made by duplicating a curve, and a singular zone can be made by duplicating a surface. The user simply needs to specify which type of singularity is being created, and enter a dimension for the singularity. Radiobuttons should also be set to generate either a default name, a specified name, or to save the new object in the object manager results buffer. When the "Create Singularity" button is hit, the program will prompt for the object to be duplicated. Since no change in the display is noticeable for this operation, no example is shown.

The text commands generated are :

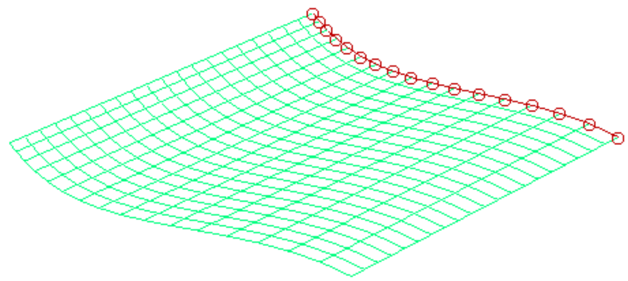
```
om select objecttype object_definition
create singularity dimension n_duplications
file save {default | name objectname}
```

5.8 Create Copy

Selection of the "Create Copy ..." submenu causes the window below to be displayed.



This option is used to create a copy of an entity. By selecting a curve from a surface and issuing a copy command, the user can extract a curve off a surface and save it as an independent object. Similarly, a point could be extracted from a curve or a surface from a zone in this manner. Although not currently possible from the GUI, in text mode the copy command can also be used to extract a subregion of an object of the same dimension to save as an independent object — that is, extract only points 5 through 10 from a curve with 20 points on it. From the GUI window, the user needs only to specify the types of objects to copy, whether to prompt for multiple objects, and specify whether to generate a default name, a specified name, or to save the new object in the object manager results buffer. When the "Make Copy" button is hit, the program prompts for the objects to be copied and generates the copies. In the image below, the red curve was copied off of the surface and saved as an independent object.

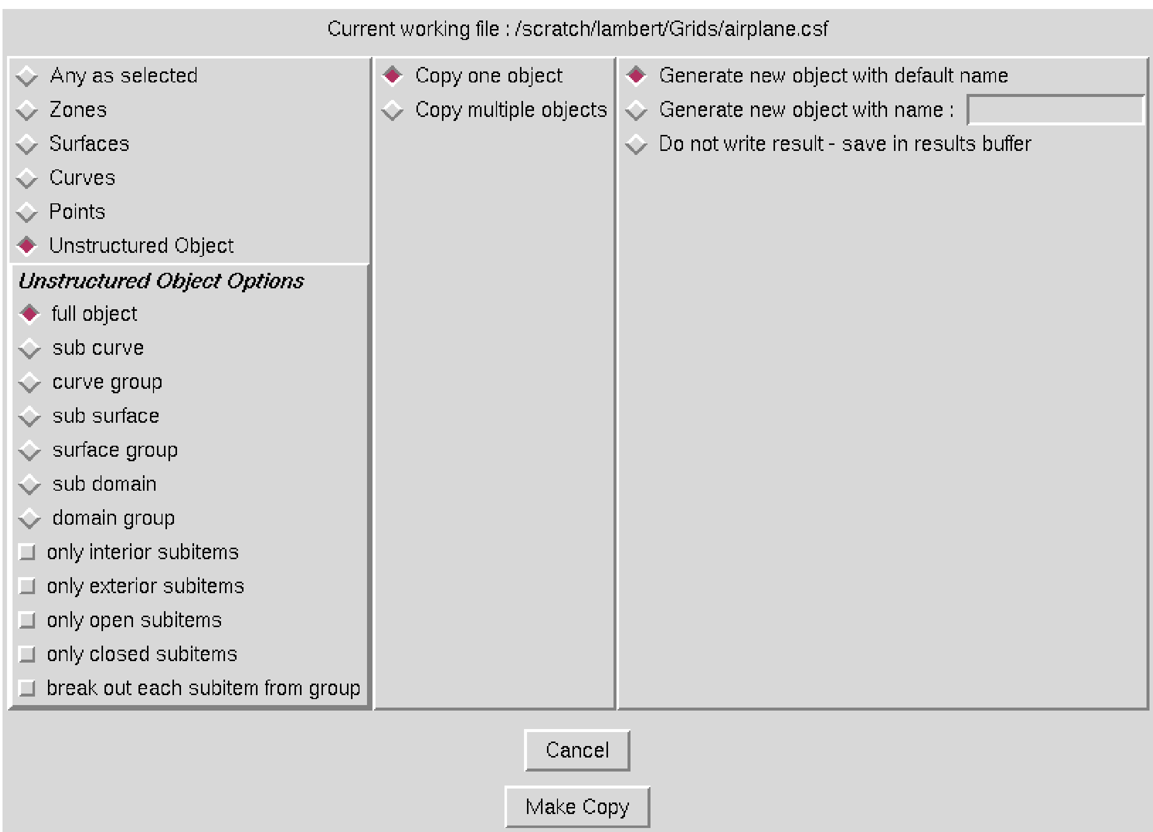


The text commands generated are :

`om select` *objecttype object_definition* (Multiple instances)
`create copy`
`file save name` *objectname*

5.8.1 Unstructured Object Options

Selection of the "Unstructured Object" radio button causes the window to display further options for creating copies of unstructured objects, as shown in the following example.

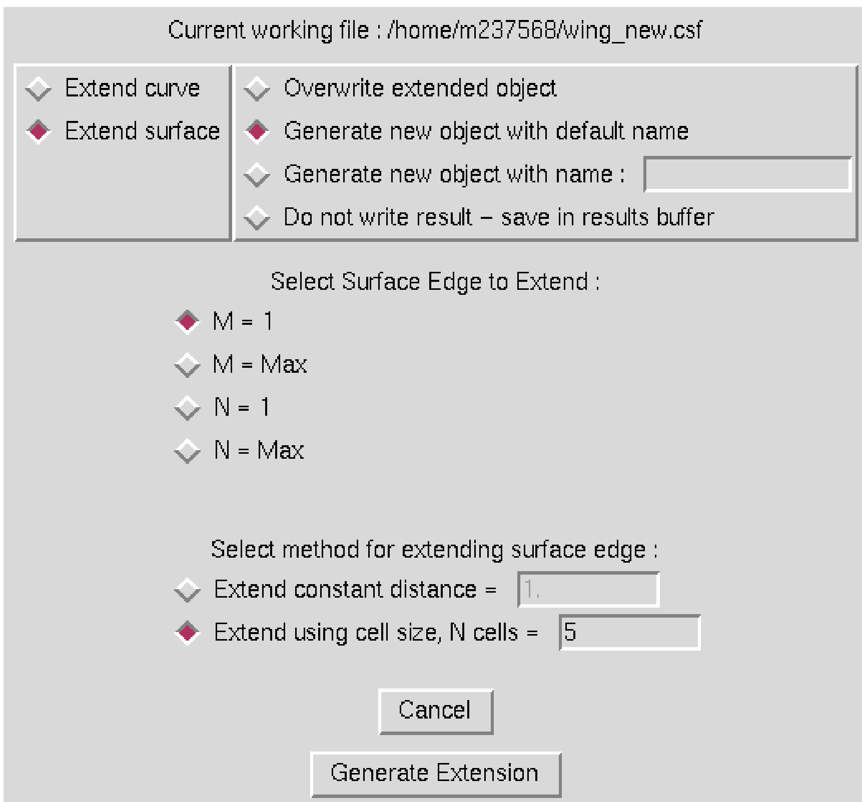


A list of unstructured sub-entities is displayed in the bottom left of the window. Here the user may select the type of sub-object to be copied out of the original unstructured object by selecting one of the radio-button boxes (diamond shape boxes). Possible selections are to copy out a sub-curve, sub-curve group, sub-surface, sub-surface group, sub-domain, or sub-domain group. In addition, the user may further restrict which items should be selected by selecting one or more modifier check boxes (square boxes). The modifiers include interior subitems, exterior subitems, open subitems, and closed subitems. Interior subitems are items that are on the interior of an object. For instance, an interior sub-curve lies on the interior of a surface mesh (not on the outer loop). An open subitem is a non-closed item. For instance, an open curve would have differing first and last points, whereas a closed curve would contain the same first and last point (forms a closed loop).

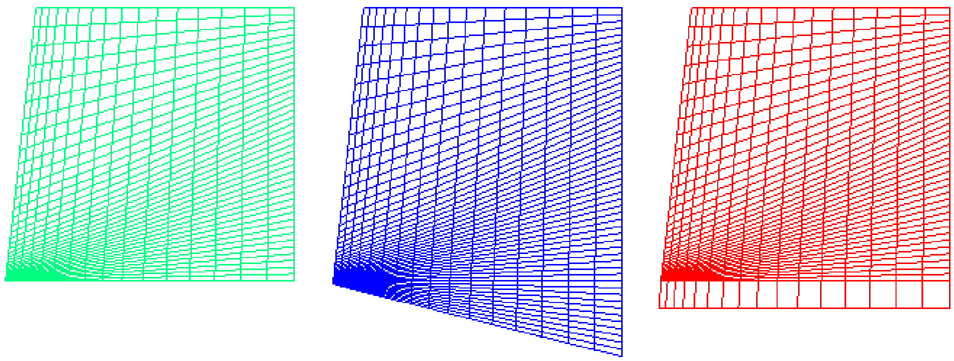
All of the sub-surfaces of an object can be copied by selecting "full object" *and* selecting the "break out each subitem from group" boxes. All of the sub-surfaces of an surface group can be copied by selecting "surface group" *and* selecting the "break out each subitem from group" boxes.

5.9 Create Extension

Selection of the "Create Extension ..." submenu causes the following window to be displayed.



This option is used to either extend one end of a curve or to extend one of the four sides of a surface. The user must set the radiobutton at the upper left to indicate whether a curve or surface is to be extended. For a curve, the user must then set whether to extend the $M = 1$ or $M = max$ end of the curve. For a surface, the $N = 1$ and $N = max$ sides can also be extended. The curve or surface is then extended by adding points to the surface in the direction defined by the cell(s) at the end of the curve or surface. There are two methods for extending a surface or curve. First, the user can simply specify a constant distance to extend the end or side. Secondly, the user can have the program simply add a specified number of cells to the end or side. For this case, the cell size at the end is duplicated as many times as requested. The user must also specify whether to generate a default name, a specified name, or to save the new object in the object manager results buffer. When the "Generate Extension" button is hit, the program prompts for the object to be extended and generates the extended surface or curve. In the image below, the red surface shows a surface edge extended a constant distance, and the blue surface shows a surface edge extended by adding 11 cells to the green surface. (Surfaces translated to show differences. Otherwise new surfaces would totally obscure original surface which exists as a subset of the new surface.)



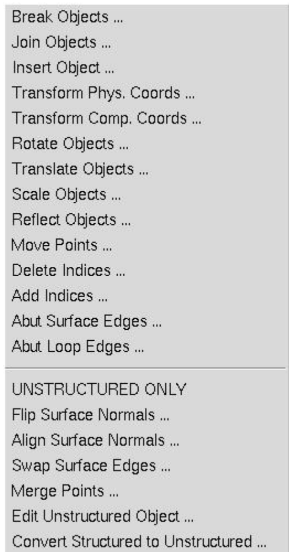
The text commands generated are :

om select *objecttype object_definition*
create extend edge {m1 | mm | n1 | nm} {distance *dist* | ncells *ncells*}
file save name *zonename*

6 Manipulate Menu

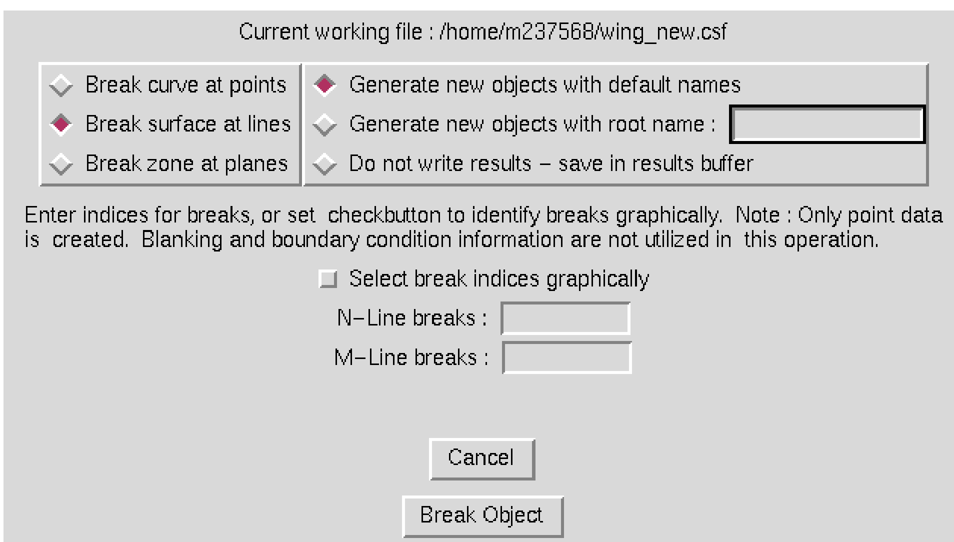
The Modular Aerodynamic Design Computational Analysis Process (MADCAP) “Manipulate” library and menus are used to create new objects such as points, curves, surfaces, and zones. Typically, objects created using this menu are derived from existing objects of the same dimension, whereas objects created by using the Create Menu (see Section 5) are created from scratch or from other points and curves displayed on screen and used as reference points.

Selection of the “Manipulate” menu item produces the drop down menu shown below.



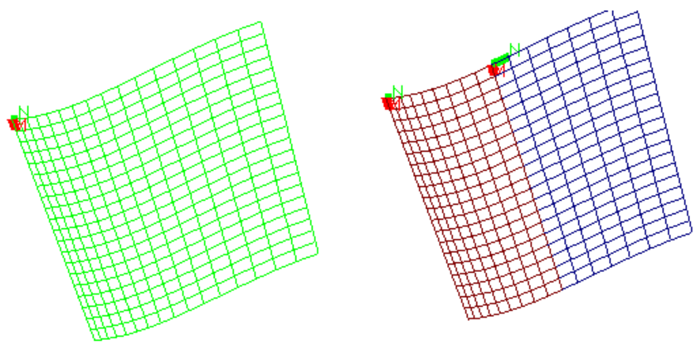
6.1 Break Objects

Selection of the “Break Objects ...” submenu causes the following window to be displayed.



This operation breaks a structured object into smaller pieces by specifying indices in the object at which to break it up, or breaks an unstructured object by specifying surfaces or surface groups in the object which should be extracted to create a new object. The window above shows the entry boxes for a surface, in which the user may enter a list of N and M lines at which to break the surface. For curves, the user is presented a single entry box in which to enter point indices to break at, while for zones, the user is presented with three entry boxes in which to enter I, J, and K indices. The user may also use the checkbutton to indicate that the break locations are to be selected graphically.

The radiobuttons at the top right should also be used to indicate whether to generate default names, use a specific name entered in the entry box, or to save the results in the result buffer without writing to file. When the "Break Objects" button is picked, the program prompts for the object to be broken, as well as the break locations if that information is to be selected graphically. The blue and red surfaces in the image below show the result of breaking the green surface at the N = 5 line. (Surfaces have been translated to show the effect of the operation — normally the new surfaces would obscure the original surface.)



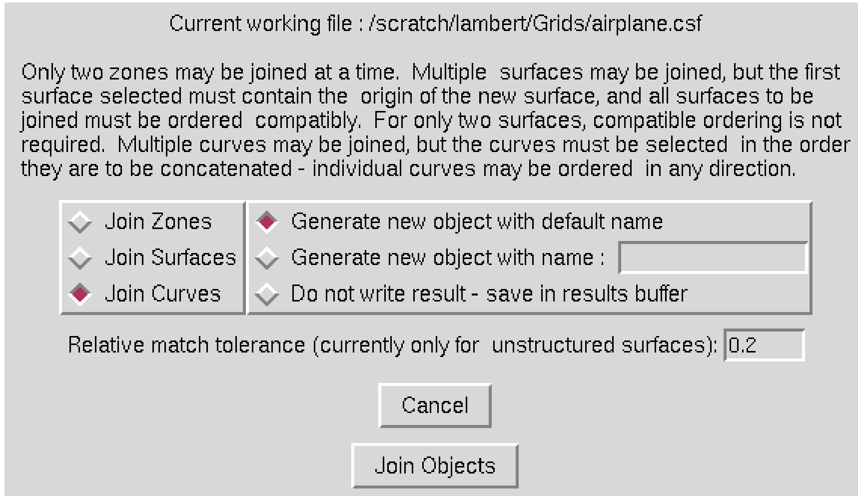
When breaking an unstructured object at a sub-surface or sub-surface group, two new objects will be created, one containing the selected sub-surfaces and another containing all of the remaining surfaces. This operation is typically used to extract a surface or group of surfaces to repair. They can later be reinserted in the full unstructured object using the join command, described in Section 6.2.

The text commands generated by this operation are:

`om select {point | curve | surface | usurface | zone}` *object_definition* (To define object to be broken)
`om select {point | curve | surface | usurface}` *object_definition* (Multiple instances, to define break locations)
`manip break index {m` *mloc* `| n` *nloc* `| i` *iloc* `| j` *jloc* `| k` *kloc*`}`
`file save name` *objectname*

6.2 Join Objects

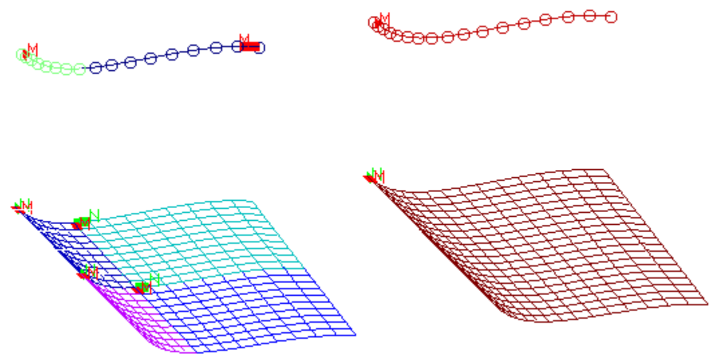
Selection of the "Join Objects ..." submenu causes the following window to be displayed.



This operation is used to combine curves, surfaces, or zones that share a common edge into a single curve, surface, or zone by eliminating the duplicated edge and creating a single new object. Only two zones may be joined at once. If only two structured surfaces are being joined, they do not need to have their N and M-family curves ordered consistently to be joined — the created surface will follow the ordering of the first surface selected. If more than two surfaces are to be joined, the surface containing the new surface origin must be selected first, and all additional surfaces must follow this orientation. For multiple curves, the order of points on the curves is not important, but the curves should be selected in the order they are to be concatenated. On the window, the user needs only to specify the type of objects to be joined and whether to generate a default name, a specified name, or save the results in the results buffer without writing to file.

When joining unstructured surfaces, the match tolerance may be used to control how closely the edges of the surfaces must match in order to be joined. The tolerance is a relative tolerance that uses a percentage of the local spacing of the boundary loop curves of the objects being joined. For instance, a value of 0.2 will join any nodes that lie within 20% of the edges on either side of the point being joined. If too large of a tolerance is specified, the surfaces may join in ways that were unintended by the user. If too small of a tolerance is specified, the surface may not join at all points. Note, the objects that are being joined must have the same number of points on either side of the join gap.

When the “Join Objects” button is picked, the program will prompt to select the objects to be joined from the display. The red surface and curve shown in the image below were generated by joining the other surfaces and curves. (Surfaces have been translated to show the effect of the operation — normally the new surface would obscure the original surfaces.)



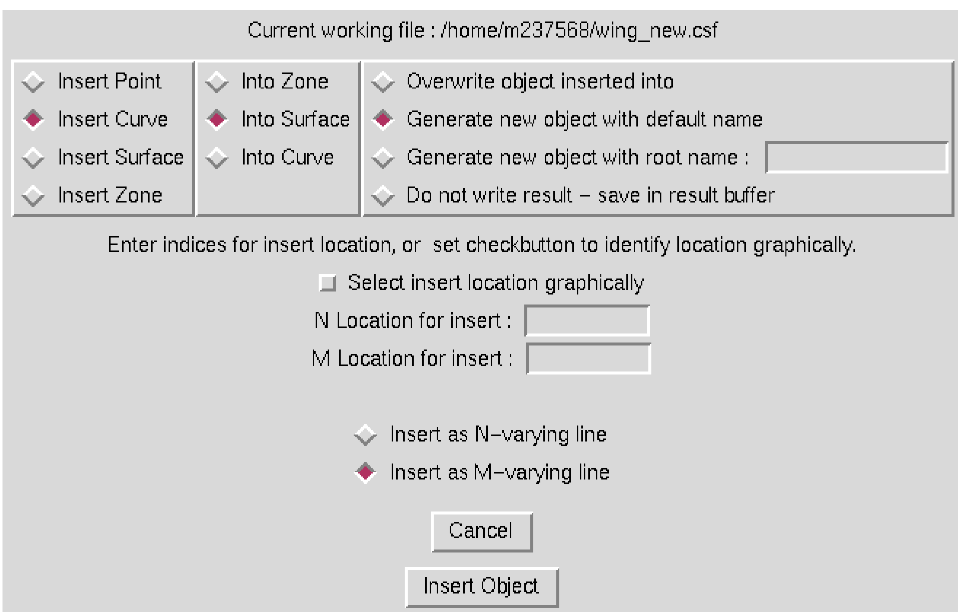
In this example, the green curve to the left was selected first, so the final curve generated is ordered as the green curve was. For the surface, the blue surface at the upper left was selected first, and the other three surfaces follow the same ordering convention. The result surface is ordered consistent with all of the selected surface, and the origin is at the origin of the blue surface at the upper left.

The text commands generated by this operation are:

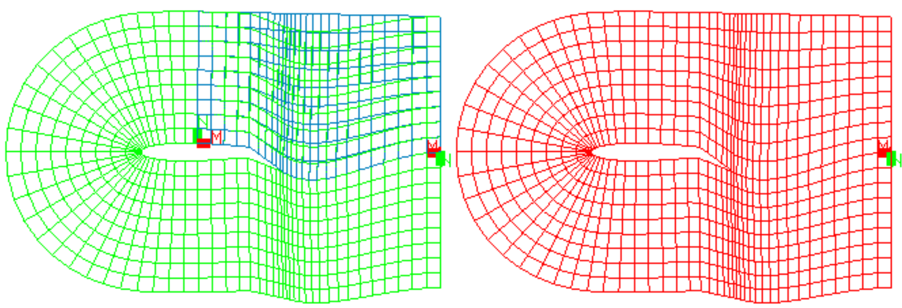
`om select {curve | surface | zone}` *object_definition* (Multiple instances, to define objects being joined)
`manip join`
`file save name` *objectname*

6.3 Insert Object (Structured Grids Only)

Selection of the "Insert Object ..." submenu causes a window similar to the following to be displayed.



This operation is used to insert the points making up one structured grid object into the definition of another structured grid object of the same or higher dimension. The coordinate values, in the same computational ordering, are used to replace the coordinate values in the target object at a specified location. When inserting a point, only the M, M and N, or I, J, and K values need to be specified for the location to insert the point in the target curve, surface, or zone. When inserting a curve into a surface or zone, the user must specify whether to insert the curve as an N-, M-, I-, J-, or K-varying curve in addition to the location to begin the insertion. When a surface is inserted into a zone, the user must specify whether it is to be inserted as an I, J, or K plane in addition to the location. The insert location may be entered at the entry boxes or picked graphically from the screen. Also, the user must set the radiobutton to indicate whether to overwrite the object being inserted into, generate a default name, use a specified name, or save the results in the result buffer. When the "Insert Object" button is picked, the program prompts first for the object being inserted into, then for the object to insert, and finally for the insert location (unless that information is included in the window entry boxes). The red surface at the bottom shows the result of inserting the blue surface into the green surface at the M = 55, N = 1 location. (The result surface has been translated to avoid overplotting on the original surfaces).

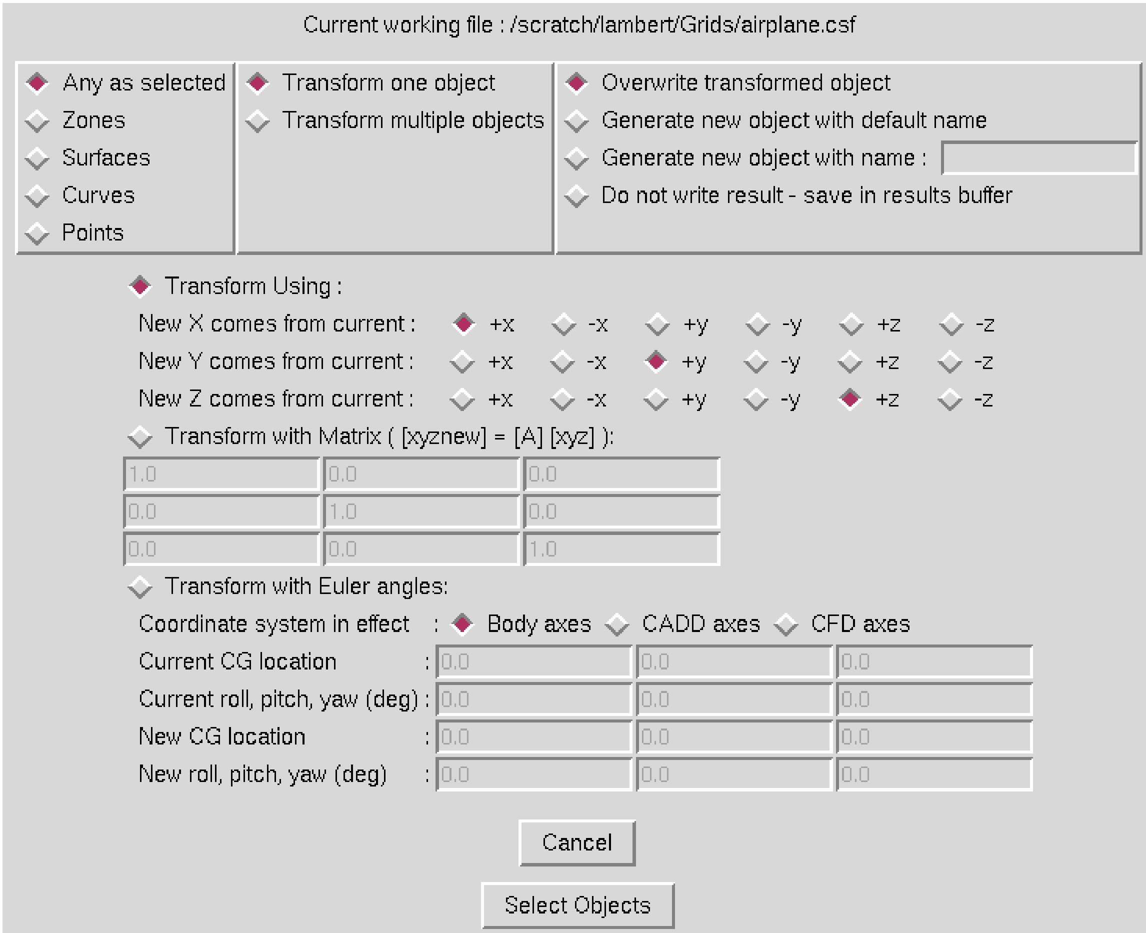


The text commands generated are:

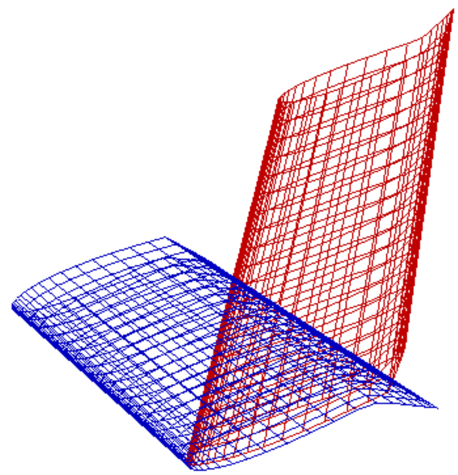
`om select {point | curve | surface | zone}` *object_definition* (To define object being inserted into)
`om select {point | curve | surface | zone}` *object_definition* (To define object being inserted)
`om select point` *object_definition* (To define insert location)
`manip insert type {point | curve | surface | zone} [m` *mloc* `| n` *nloc* `| \`
`i` *iloc* `| j` *jloc* `| k` *kloc*`]`
`file save {default | same | name` *curvename*`}`

6.4 Transform Phys. Coords

Selection of the "Transform Phys. Coords" submenu causes the window shown below to be displayed.



This operation is used to transform the physical (x, y, z) coordinates of an object. The user must first set the radiobuttons at the upper left of the window to identify the types of objects that are to be selected from the graphics window and whether a single object or multiple objects are to be transformed. Next, the radiobuttons identifying whether a full transformation matrix is to be entered or a simple exchange of axes is to be performed. For an exchange of axes, any coordinate direction can be transformed into any new coordinate direction. If a transformation matrix is entered, it pre-multiplies the coordinate data to create new coordinate data. The user should also select a radiobutton to indicate whether the object generated should overwrite the transformed object, generate a new default name, a specified name, or save the object in the result buffer. In the image below, the red surface shows the result of transforming the z coordinate of the blue surface into "y" and the y coordinate into "$-z$".

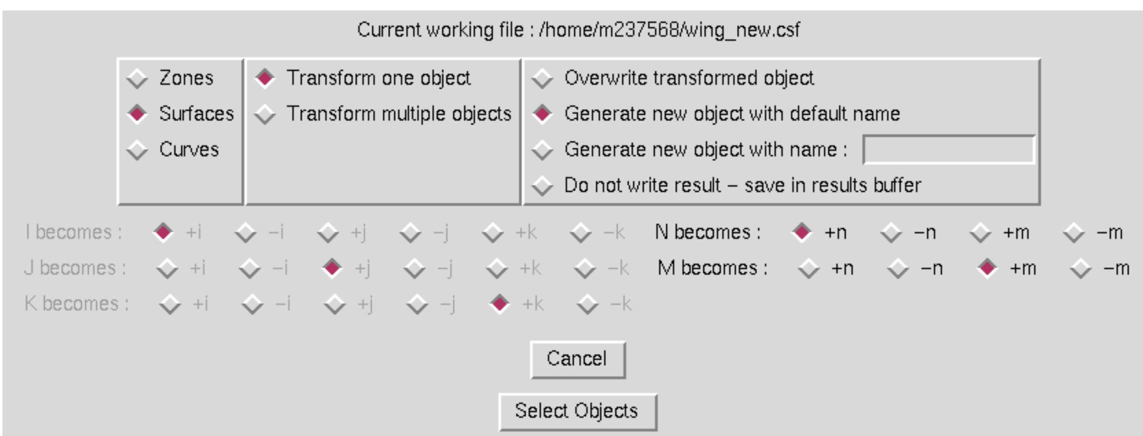


The text commands generated are:

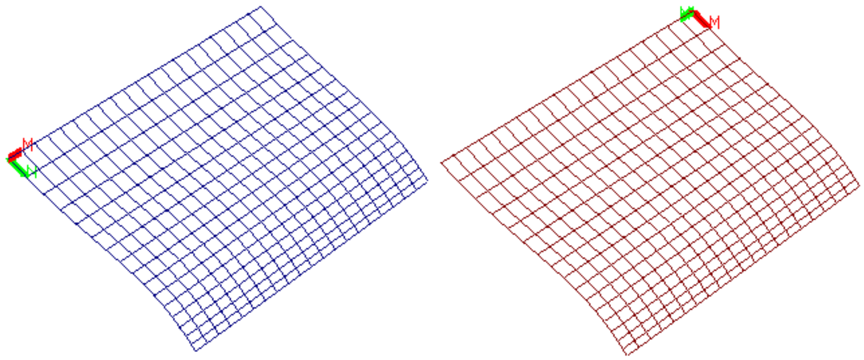
`om select {zone | surface | curve | point}` *object_definition* (Multiple instances)
`manip transform {xyz` *xyzt xyzt xyzt* `| matrix` *m11 m12 m13 m21 m22 m23* `\`
m31 m32 m33`}`
`file save {default | name` *surfacename* `| overwrite}`

6.5 Transform Comp. Coords (Structured Grids Only)

Selection of the "Transform Comp. Coords ..." submenu causes the following window to be displayed.



This operation is used to transform the computational coordinates of an object. For zones, the user specifies a new computational assignment for each of I, J, and K. For surfaces, the computational indices are N and M. For curves, the only transformation possible is to switch the direction of points on the curve. The results of computational transformations are only visible if the Highlight Object Origins mode is set (see Section 4.11), since there is no change to the data itself, only to its organization. The user should specify the type of object to transform, whether one or multiple objects or to be transformed, and how to save the result object. The red surface in the image below shows the result of transforming the N direction of the blue surface into +M, and the M direction into −N. (The result object has been translated to avoid overplotting the original object).

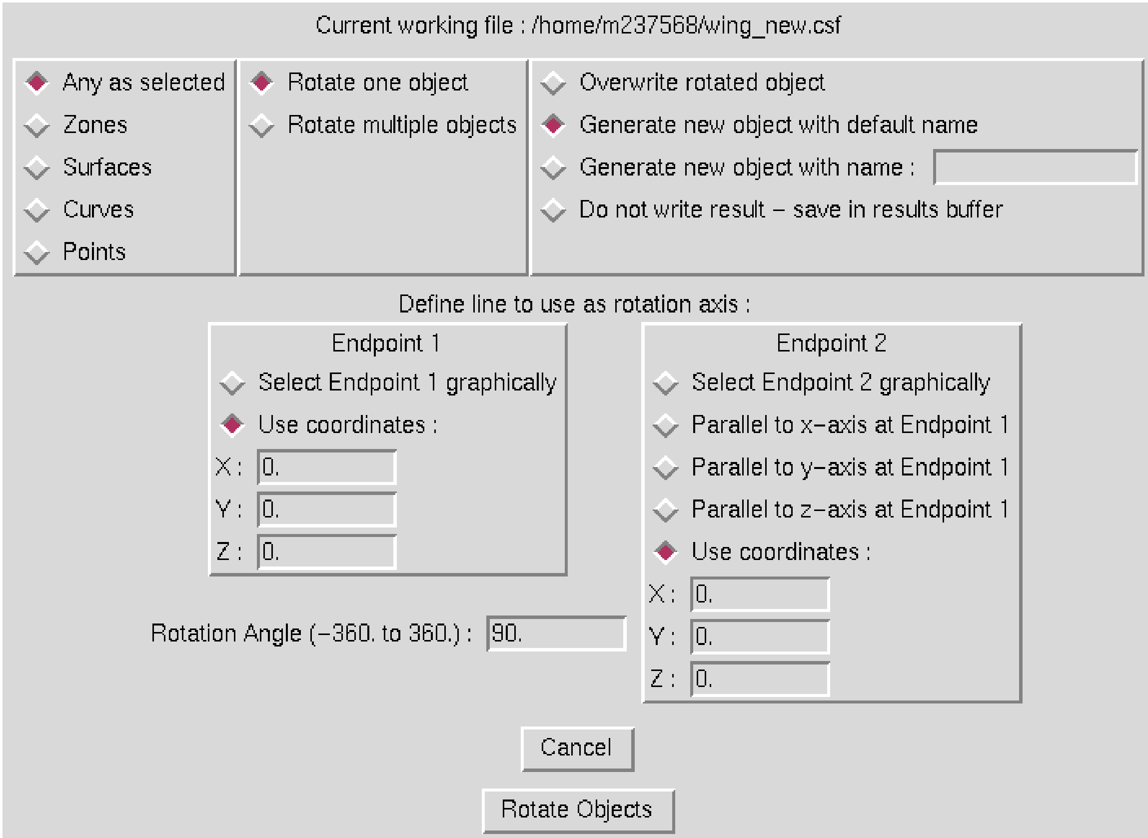


The text commands generated are:

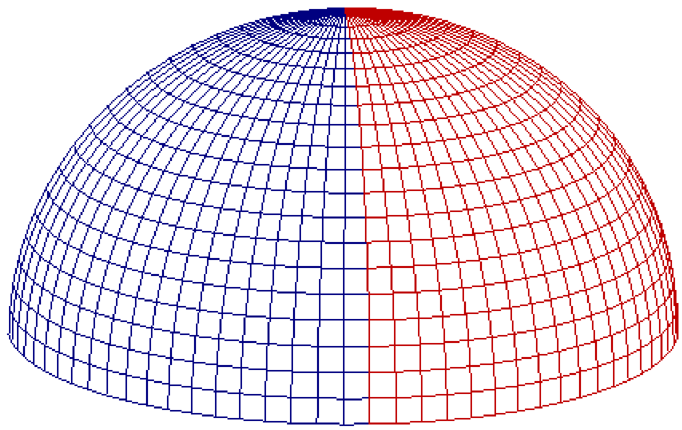
`om select {zone | surface | curve | point}` *object_definition* (Multiple instances)
`manip transform {ijk` *ijkt ijkt ijkt* `| nm` *nmt nmt*`}`
`file save {default | name` *surfacename* `| overwrite}`

6.6 Rotate Objects

Selection of the "Rotate Objects ..." menu causes the following window to be displayed.



This operation is used to rotate an object a given angular distance about an arbitrary line. Any type of object can be rotated, and the user can specify that either a single object or multiple objects be rotated at once. The two endpoints for the rotation line can either be selected graphically or keyed in on the window. Optionally, rather than a second endpoint being specified, the object can be rotated about a line parallel to one of the principal axes at the first endpoint. The amount of angular rotation, between −360 and 360 degrees, should be entered in the entry box on the bottom left. The user must also select a radiobutton indicating the results sould overwrite the original object, be stored with a default name, a specified name, or saved in the results buffer. When the "Rotate Objects" button is selected, the user will be prompted for the objects to rotate and the endpoints of the rotation line if they are to be seleted graphically. In the image below, the red surface has been generated by rotating the blue object 90 degrees about the vertical axis at the singular edge at the top of the object.

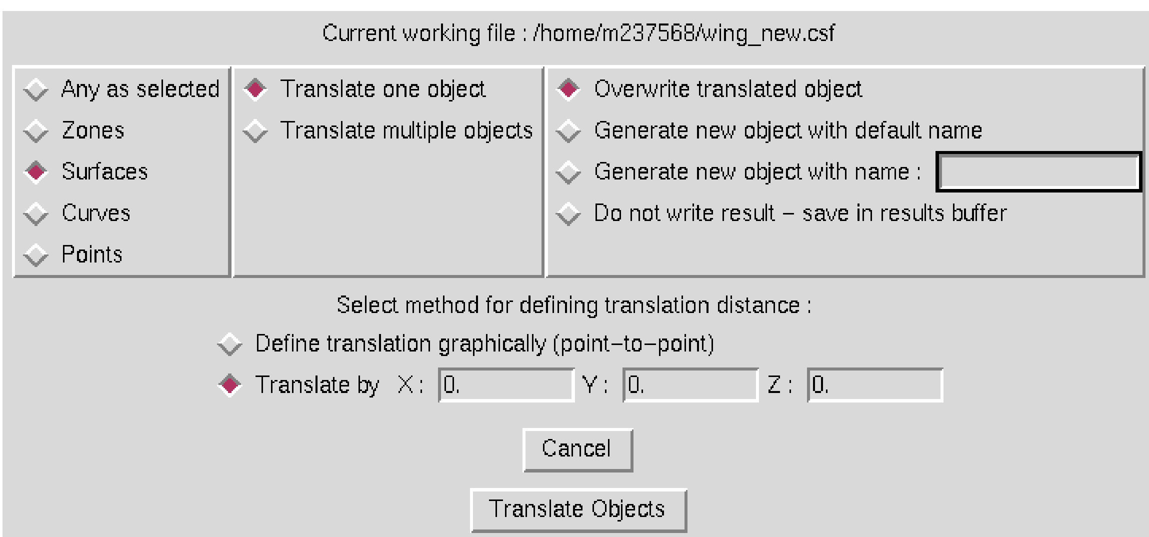


The text commands generated are:

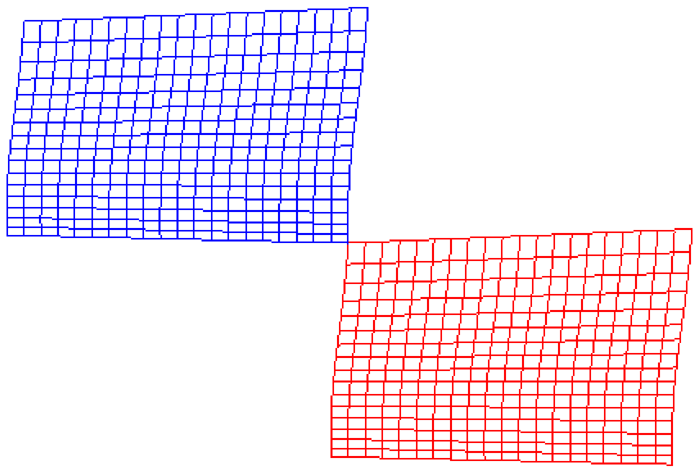
`om select {zone | surface | curve | point}` *object_definition* (Multiple instances)
`om select point` *point_definition* (One or two instances, to define rotation)
`manip rotate parallel_to {x | y | z | line} angle` *angle*
`file save {default | name` *objectname* `| overwrite}`

6.7 Translate Objects

Selection of the "Translate Objects ..." submenu causes the window below to be displayed.



This operation is used to translate an object a given distance. Any object type can be translated, and a single object or multiple objects may be translated at once. The translation can be entered as x, y, and z translation components or by graphically selecting two points on the screen to define the translation vector. The user must also select a radiobutton indicating the results sould overwrite the original object, be stored with a default name, a specified name, or saved in the results buffer. When the "Translate Objects" button is selected, the user will be prompted to select the objects to translate, and for the points defining the translation vector if this data is to be entered graphically. In the image below, the red surface resulted from a translation of the blue surface. The translation vector was defined by selecting the upper left corner of the surface as the vector origin and the lower right corner of the surface as the vector termination.

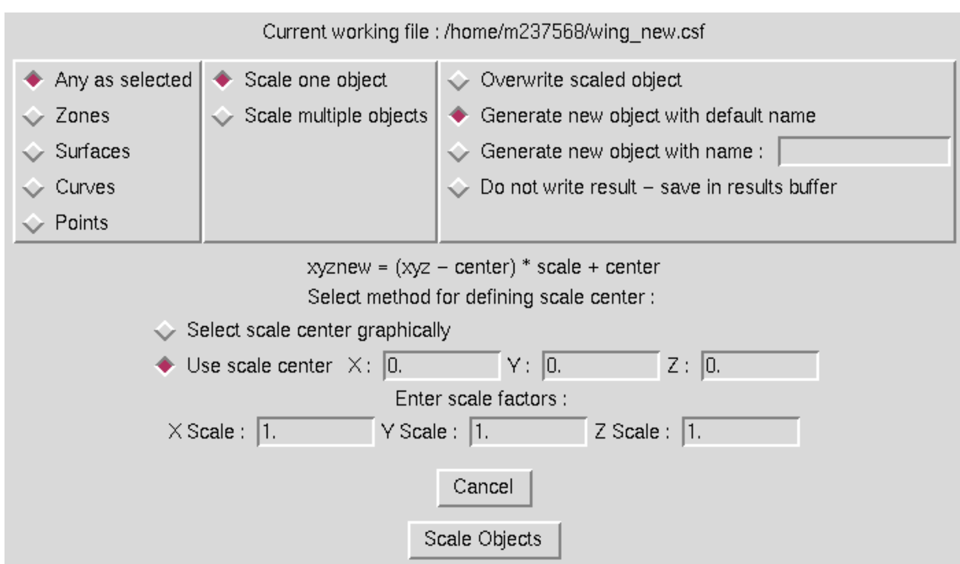


The text commands generated are:

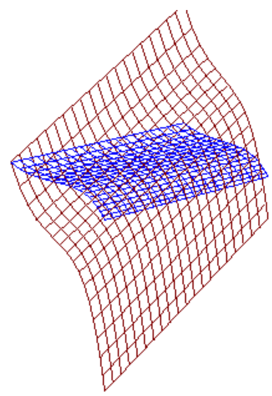
`om select {zone | surface | curve | point}` *object_definition* (Multiple instances)
`om select point` *point_definition* (Two instances, to define translation vector)
`manip translate [x` *x* `y` *y* `z` *z*`]`
`file save {default | name` *objectname* `| overwrite}`

6.8 Scale Objects

Selection of the "Scale Objects ..." submenu causes the window below to be displayed.



This option is used to create a scaled version of an object. The scale factors are applied after subtracting off a "scale center" from the coordinates. This center is then added back on after the scale factors have been applied. Any object type can be scaled, and a single object or multiple objects can be scaled at once. The scale center can be defined either in the entry boxes in the window or selected graphically. The user should enter the desired scale factors for each coordinate direction in the entry boxes at the bottom. By entering non-uniform scale factors, objects may be stretched or compressed in addition to being simply scaled. The user must also select a radiobutton indicating the results sould overwrite the original object, be stored with a default name, a specified name, or saved in the results buffer. When the "Scale Objects" button is selected, the user will be prompted to select the objects to be scaled, and for a scale center if this is to be defined graphically. The red surface in the image below was generated by scaling the blue surface by 0.5 in the x direction, 1.0 in the y direction, and 2.0 in the z direction, with a scale center located at the upper left corner of the surface.

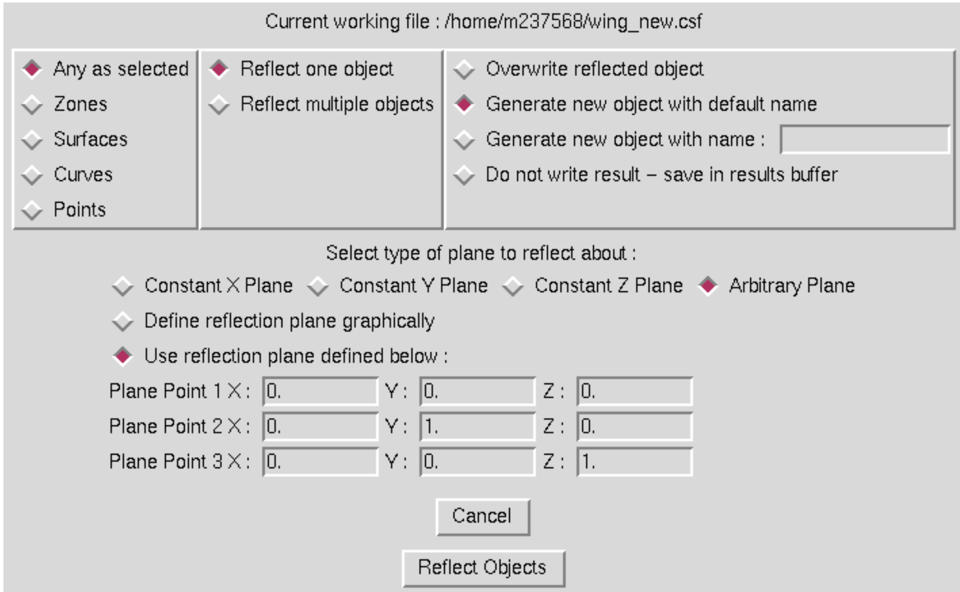


The text commands generated are:

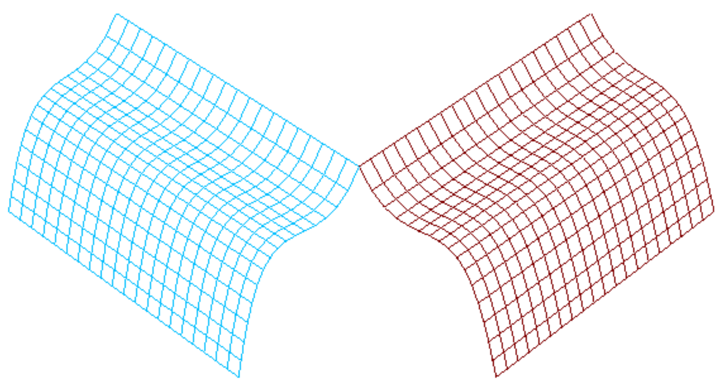
`om select {zone | surface | curve | point}` *object_definition* (Multiple instances)
`om select point` *point_definition* (To define scale center)
`manip scale {x` *xscale* `y` *yscale* `z` *zscale*`}`
`file save {default | name` *objectname* `| overwrite}`

6.9 Reflect Objects

Selection of the "Reflect Objects ..." submenu causes the following window to be displayed.



This option is used to reflect an object about any arbitrary plane. Any object type can be reflected, and a single object or multiple objects can be reflected at once. The user must specify on the window whether to reflect about a constant x, y, or z plane or a completely arbitrary plane. For a reflection about a constant x, y, or z plane, only one point is required to define the reflection plane, while three points are required for an arbitrary plane. These points may be defined in the window or selected graphically. The user must also select a radiobutton indicating the results sould overwrite the original object, be stored with a default name, a specified name, or saved in the results buffer. When the "Reflect Objects" button is selected, the user will be prompted for the objects to reflect, and for the point(s) defining the reflection plane if they are to be defined graphically. The red surface in the image below is the reflection of the blue surface about a constant vertical plane located at the upper right corner of the blue surface.

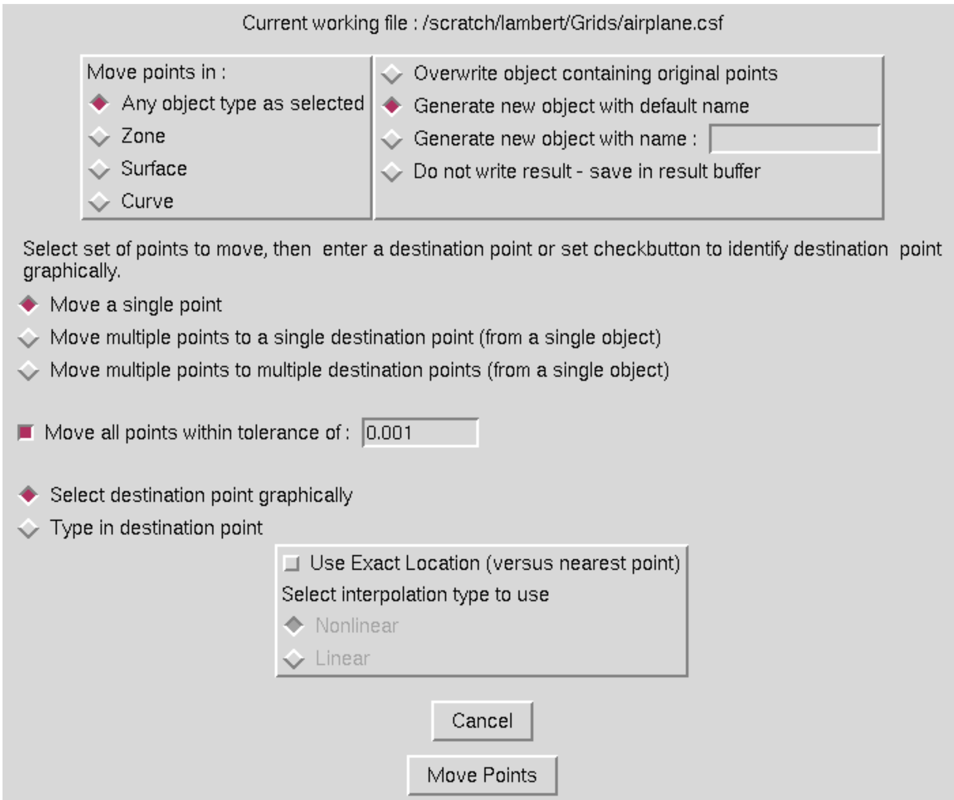


The text commands generated are:

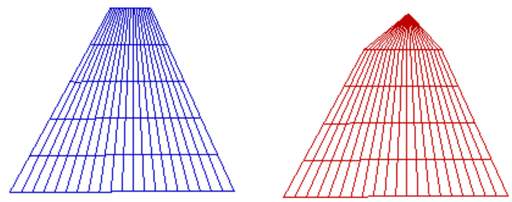
`om select {zone | surface | curve | point}` *object_definition* (Multiple instances)
`om select point` *point_definition* (One to three instances, to define reflection plane)
`manip reflect {x | y | z | arb}`
`file save {default | name` *objectname* `| overwrite}`

6.10 Move Points

Selection of the "Move Points ..." submenu causes the following window to be displayed.



This option is used to move points that are members of any object type to a new coordinate location while leaving the other points on the object unchanged. Multiple points on an object may be selected to be moved. Also, an option may be set such that all points withing a given tolerance from a selected point should be moved. The destination point can either be selected graphically or keyed in on the window. The user must also select a radiobutton indicating the results sould overwrite the original object, be stored with a default name, a specified name, or saved in the results buffer. When the "Move Points" button is selected, the user will be prompted for the object containing the points to move, and then for the points themselves. If the destination point is to be selected graphically, the program will then prompt for that point as well. In the image below, the red surface shows the result of moving all of the points on the top edge of the surface to a single point to generate a singular edge on the surface. (The result surface has been translated to avoid overplotting the original surface.)

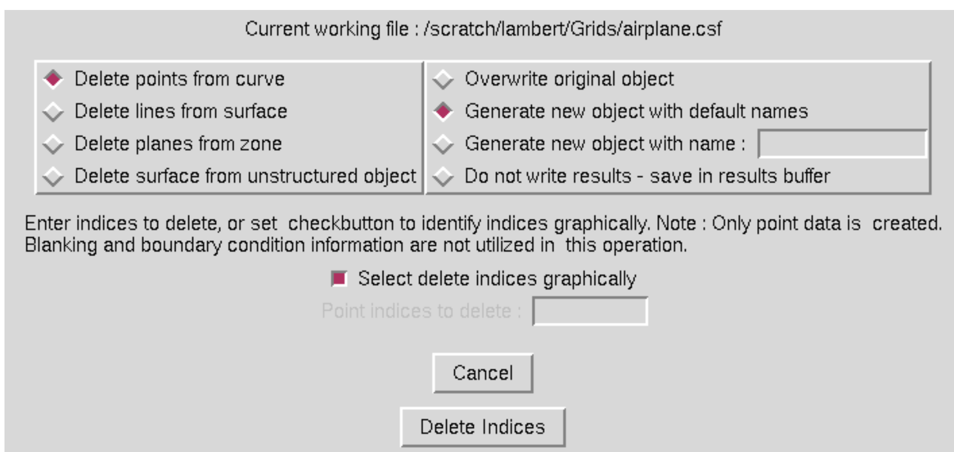


The text commands generated are:

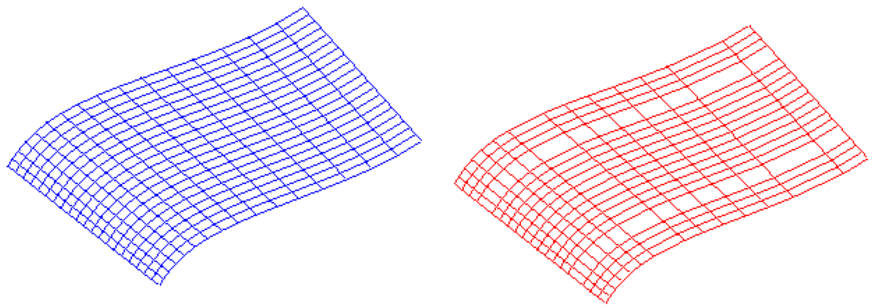
`om select point` *point_definition* (Multiple instances, to define points to be moved and destination point)
`manip movepoint [tol` *tolerance*`]`
`file save {default | name` *objectname* `| overwrite}`

6.11 Delete Indices

Selection of the "Delete Indices ..." submenu causes the following window to be displayed.



This option is used to delete specific points from a curve, lines from a structured surface, planes from a structured zone or surfaces from an unstructured object. The option for the type of delete should be set on the radiobuttons at the upper left. The specific entities to delete from an object may then be selected graphically or by entering the indices on the entry boxes. The example above shows the window when the "delete lines from surface" option is selected. The entry boxes are modified for the other two options to allow deletion of the appropriate type of indices. The user must also select a radiobutton indicating whether the result should be stored with a default name, a specified name, or saved in the results buffer. When the "Delete Indices" button is selected, the user will be prompted for the object containing the indices to delete, and then for the entities to be deleted themselves if they are to be specified graphically. In the image below, the red surface shows the result of deleting two N-lines and two M-lines from the blue surface. (The result surface has been translated to avoid overplotting the original surface.)

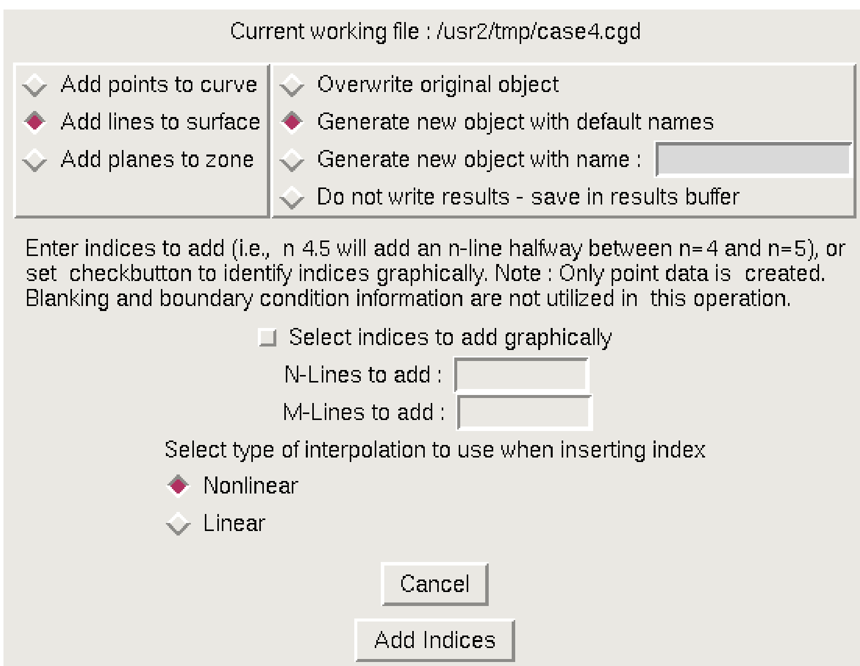


The text commands generated are:

`om select {curve | surface | zone}` *object_definition*
`om select {point | curve | surface}` *object_definition* (Multiple instances, to define indices to be deleted)
`manip delete [I` *indices*`] [J` *indices*`] [K` *indices*`] [M` *indices*`] [N` *indices*`]`
`file save {default | name` *objectname*`}`

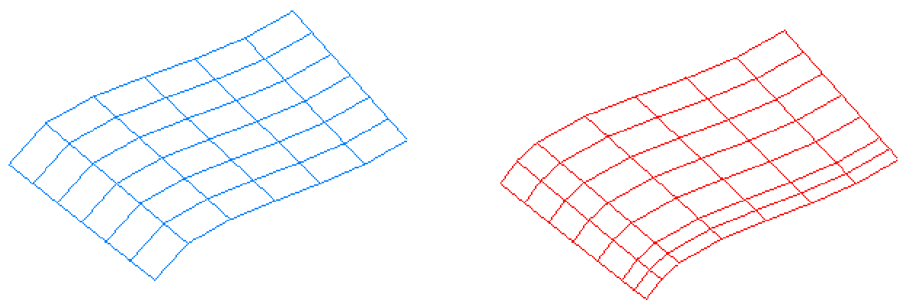
6.12 Add Indices (Structured Grids Only)

Selection of the "Add Indices ..." submenu causes the following window to be displayed.



This option is used to add points to a curve, lines to a structured surface, or planes to a structured zone. The option for the type of addition should be set on the radiobuttons at the upper left. The specific locations to add indices to an object may then be selected graphically or by entering the indices on the entry boxes. Locations in text mode are defined by a real number in index coordinates. For example, to add a new N-line halfway between N = 5 and N = 6, an add location of N = 5.5

should be specified. The window above shows the entry boxes when the "add lines to surface" option is selected. The entry boxes are modified for the other two options to allow addition of the appropriate type of indices. Since coordinate data will be added at the specified location, the user must also indicate whether the interpolation on the surface should be perfromed linearly or nonlinearly. The user must also select a radiobutton indicating whether the result should be stored with a default name, a specified name, or saved in the results buffer. When the "Add Indices" button is selected, the user will be prompted for the object to be added to and then for the locations to add indices if they are to be defined graphically. In the image below, the red surface shows the result of adding an N-line and an M-line to the blue surface between the first two indices in each direction. (The result surface has been translated to avoid overplotting the original surface.)

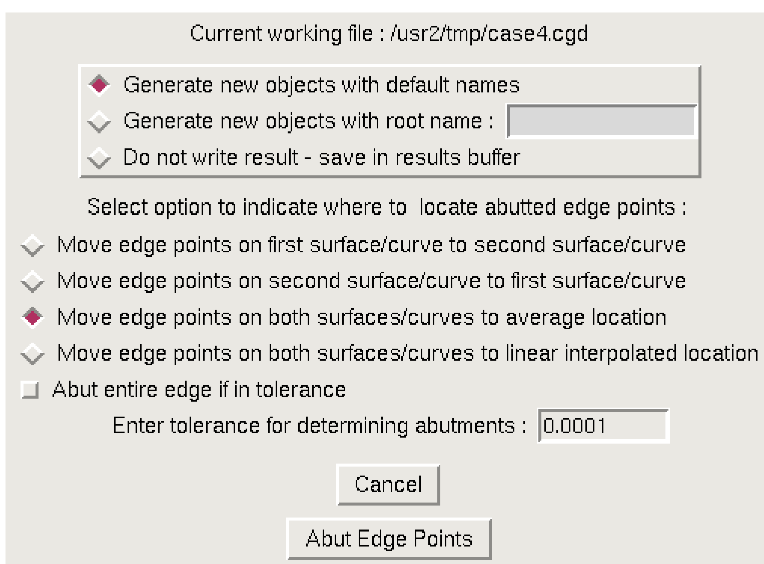


The text commands generated are:

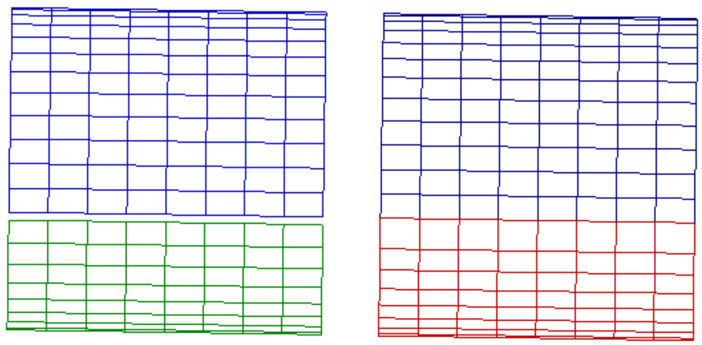
om select {curve | surface | zone} *object_definition*
manip add {[I *indices*] [J *indices*] [K *indices*] [M *indices*] [N *indices*]} \
  interp {linear | nonlinear}
file save {default | name *objectname*}

6.13 Abut Surface Edges (Structured Grids Only)

Selection of the "Abut Surface Edges ..." submenu causes the following window to be displayed.



This option is used to force the edges of two structured surfaces to abut by moving the points on the edge of one surface to match the points on the edge of a second surface, or by moving the points on both edges to an average location between the two edges. A user-defined tolerance is used to assist in locating the two abutting edges. Edges separated by more than this tolerance will not be detected as possibly abutting. The desired tolerance is entered in the entry box at the bottom of the window. The user must also select a radiobutton indicating whether the result should be stored with a default name, a specified name, or saved in the results buffer. When the "Abut Edge Points" button is selected, the user will be prompted for the two surfaces to be abutted. In the image below, the red surface shows the result of abutting the green surface to the blue surface. (The result surfaces have been translated to avoid overplotting the original surfaces.)



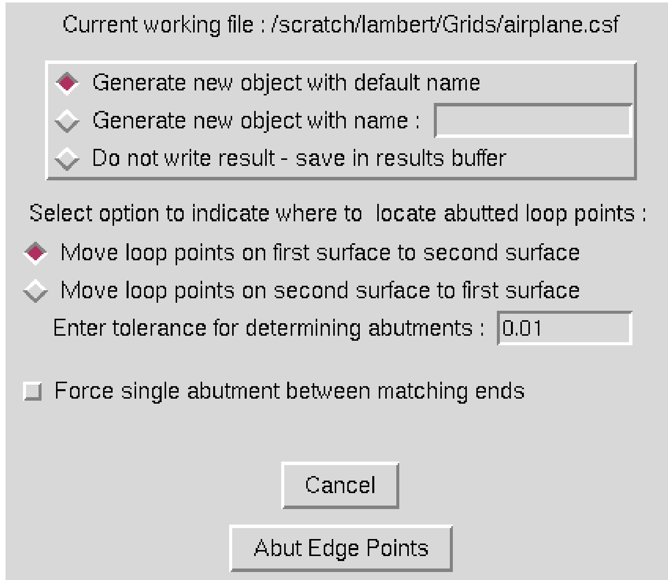
For unstructured surfaces, see the Abut Loop Edges command in Section 6.14.

The text commands generated are:

`om select surface` *surface_definition* (Two instances)
`manip abut mode {first | second | average} tolerance` *tol*
`file save {default | name` *surfname*`}`

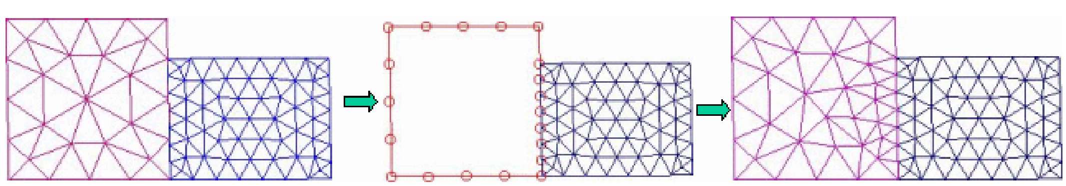
6.14 Abut Loop Edges

Selection of the "Abut Loop Edges ..." submenu causes the following window to be displayed.



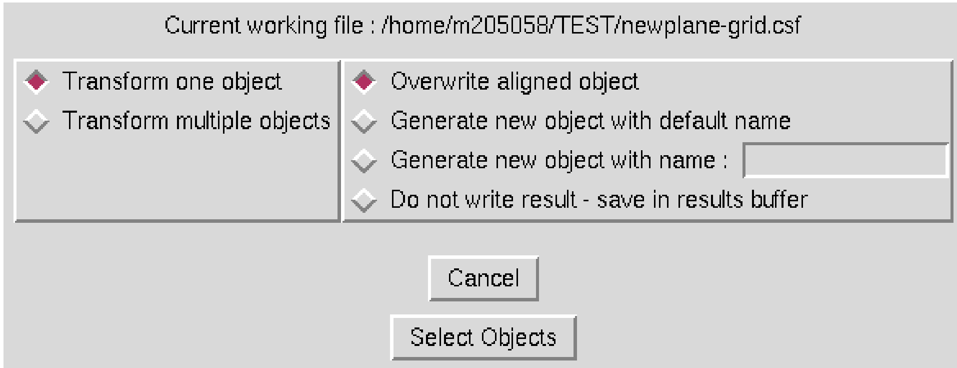
This option is used to force the edges of two unstructured surface boundary loops to abut by moving the points on one loop to match the points on the of a second loop. A user-defined tolerance is used to assist in locating the corners of the two abutting edges. The nodes along one loop are projected to the second loop and the distance between the original node and the projected node are compared against the tolerance. Once the two corners of the common section between the two loops is determined, the nodes on one loop between the corner points are replaced with the nodes from the second loop. The desired tolerance is entered in the entry box at the bottom of the window. The "Force single abutment between matching ends" button will force the abut operation to merge all of the points between the two corners of the matching region regardless of the distance of these points. In this case, the tolerance will only be used to determine the corner points.

The user must also select a radiobutton indicating whether the result should be stored with a default name, a specified name, or saved in the results buffer. When the "Abut Edge Points" button is selected, the user will be prompted for the two boundary loops to be abutted. Note that the boundary loops can be directly selected from the unstructured surface mesh and do not need to be extracted first. In the image below, the boundary loop from the red surface is abutted to the boundary loop of the blue surface. The resulting boundary loop is shown in the middle picture. The picture on the right shows the result after using the new boundary loop to make a new unstructured surface.

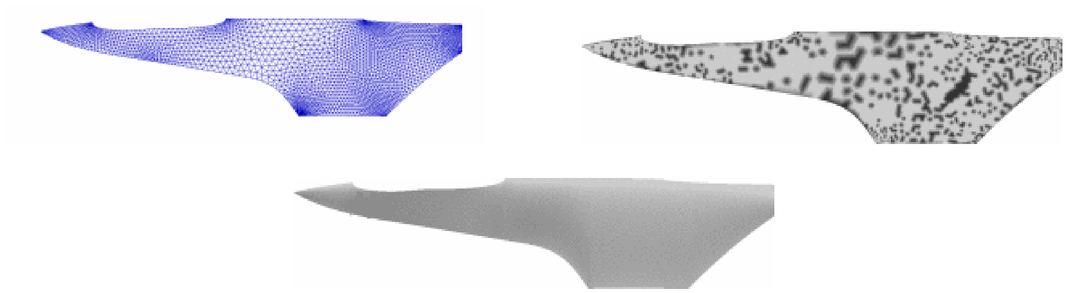


6.15 Flip/Align Surface Normals (Unstructured Grids Only)

Selection of the "Align Surface Normals ..." or the "Flip Surface Normals" submenu causes the following window to be displayed.

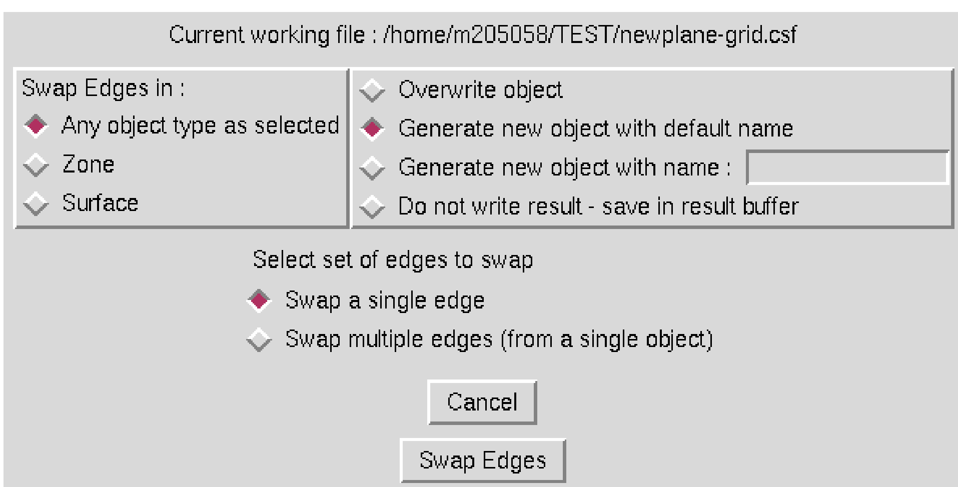


This option is used to align the normal direction of all faces in a surface to point in a consistent direction. Surfaces created within MADCAP will already have this property. This capability is useful for fixing surfaces that are imported from other programs. In the image below, the blue surface mesh has non-aligned normal vectors as evident from the shaded image to its right. After aligning the surface normals, the shaded surface is shown at the bottom.

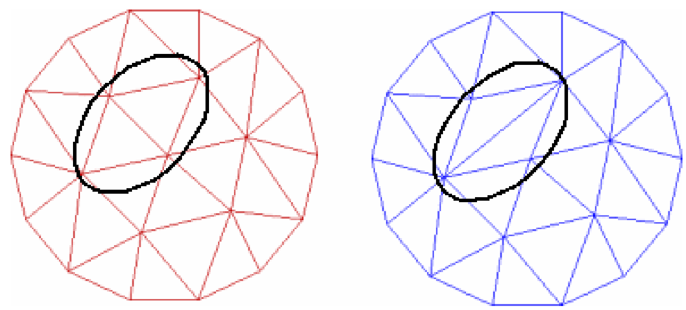


6.16 Swap Surface Edges (Unstructured Grids Only)

Selection of the "Swap Surface Edges ..." submenu causes the following window to be displayed.

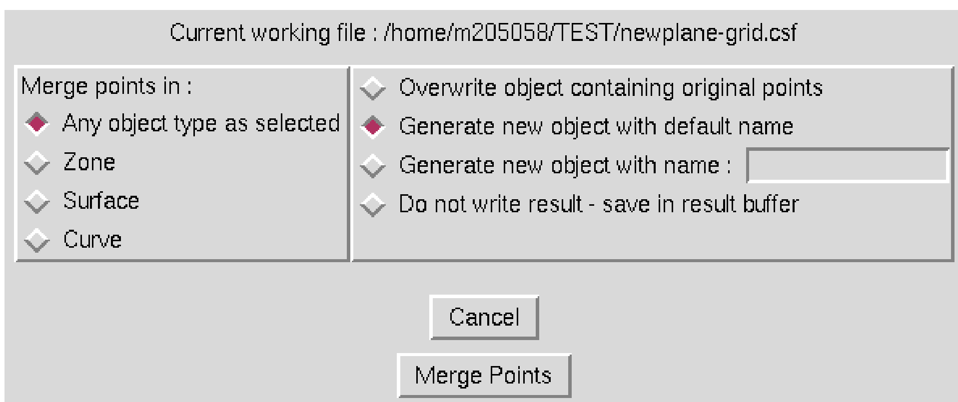


This function allows the edge between two neighboring triangles to be swapped. The user must specify whether to swap edges in a surface or a zone or either as selected, and also whether to swap a single edge or to swap multiple edges. The user must also select a radiobutton indicating whether the result should be stored with a default name, a specified name, or saved in the results buffer. When the "Swap Edges" button is picked, the user will be prompted to select an unstructured object, and then to select an edge or edges to swap. Note that if adjacent edges are to be swapped, they should be swapped one at a time. In the figure below, a single edge (highlighted by the circle) has been swapped. The resulting object is shown on the right.

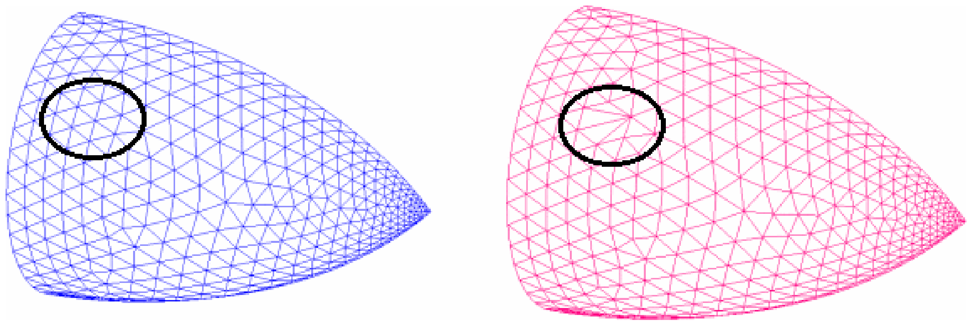


6.17 Merge Points (Unstructured Grids Only)

Selection of the "Merge Points ..." submenu causes the following window to be displayed.

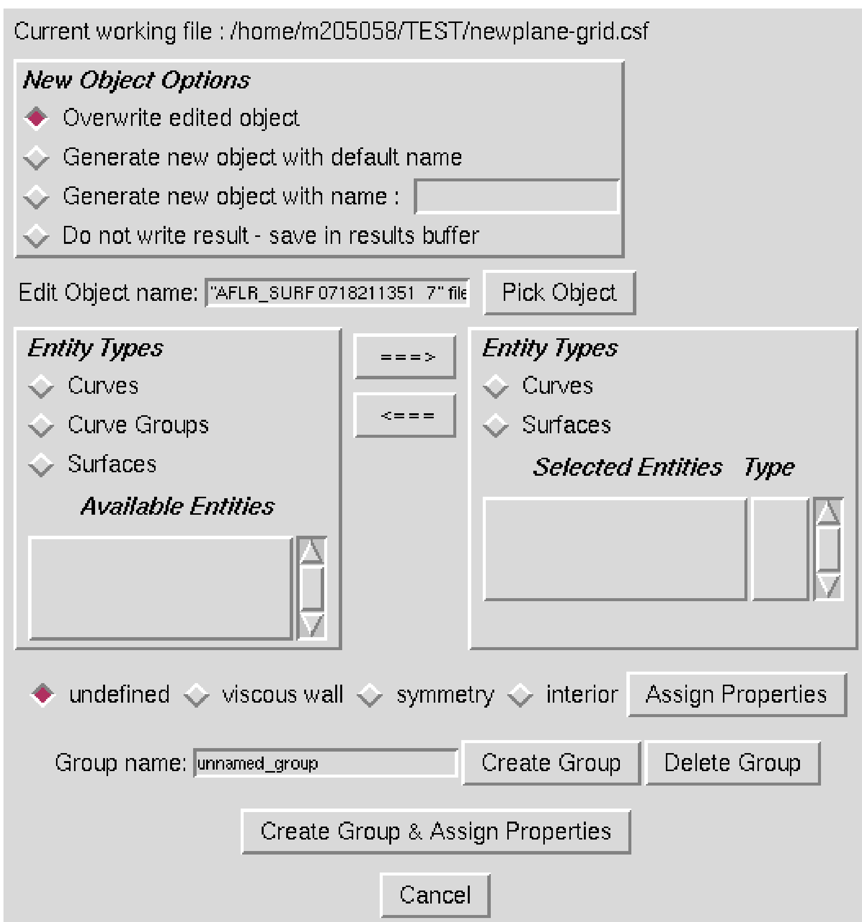


This function allows a node within an unstructured object to be combined with another node. Any degenerate faces or edges that result from this operation are automatically removed from the resulting object. Duplicate nodes are also removed. Although similar to the move nodes function, merge nodes should always be used when combining nodes within an unstructured object. The user must select the type of object to merge nodes within. The user must also select a radiobutton indicating whether the result should be stored with a default name, a specified name, or saved in the results buffer. When the "Merge Points" button is picked, the user will be prompted to select an unstructured object. The user will then be prompted to select pairs of nodes for merging. The first node in each node pair will be moved to the second node. In the image below, the two nodes highlighted by the circle have been merged. The resulting object is shown on the right.



6.18 Edit Unstructured Object (Unstructured Grids Only)

This menu allows the user to define surface or curve groups in an unstructured object or to assign properties (such as boundary conditions for the unstructured volume grid generation) to surfaces, curves or surface or curve groups. When entering this menu, first you must select the unstructured object to edit by left clicking on the "Pick Object" button. After selecting and unstructured object from the graphics display or file list, the following window will be displayed.



All of the available curves, surfaces, curves groups, surface groups in the selected unstructured object are shown on the left side of this menu under the “Entity Types” box. By selecting one of the radio buttons, all of the curves, curve groups, surfaces, or surface groups in the object can be listed. Before defining a new group or assigning properties, the entities that are to be operated on should be selected by moving them into the “Entity Types” Selected Entities list on the right of the form. To do this, highlight the surface/curves in the “Available Entities” list on the left side of the form and then select a radio button of the type of entity to select on the right side of the form. Left click on the ===> button to move the selected entities to the “Selected Entities” list on the right of the form.

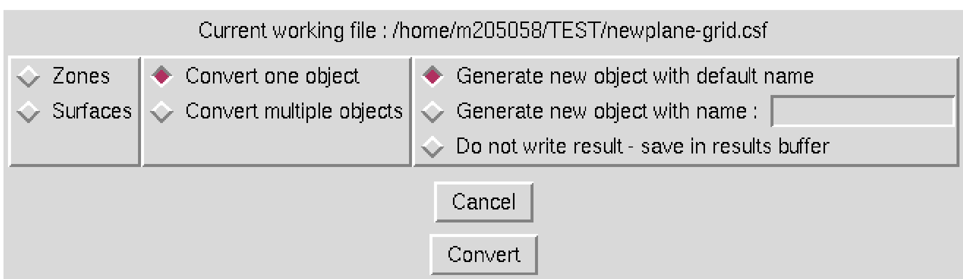
Once the desired entites are in the “Selected Entities” list, you can Assign boudary conditions properites by selecting one of the radio buttons “undefined, viscous wall, symmetry or interior” and then left click on the “Assign Properties” button.

Groups can be defined by entering a new name in the “Group name” text entry box and then left clicking on the “Create Group” button. You can assign properties and define a group at the same time by left clicking on the “Create Group & Assign Properties” button.

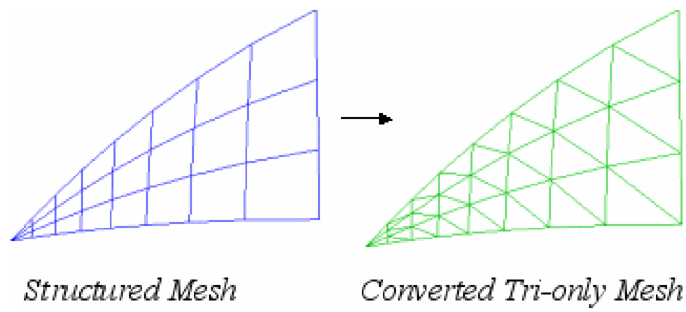
Curve or Surface groups can be deleted by selecting the curve or surface group(s) to be deleted in the “Available Entities” list and then clicking on the “Delete Group” button.

6.19 Convert Structured to Unstructured

Selection of the “Convert Structured to Unstructured ...” submenu causes the following window to be displayed.



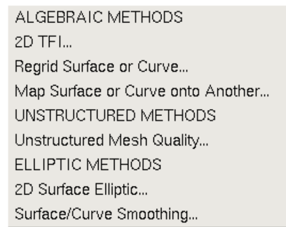
This function allows a structured surface or volume mesh to be converted to an unstructured mesh. The user must select whether to convert a surface or volume mesh. If a surface mesh is selected, then the user has the option to convert faces directly (leave quad faces as quads), or to convert all faces to triangular faces. If a volume mesh is selected, the user has the option to convert all cells to tetrahedral elements or to leave cells as they are. Note that degenerate cells or faces are converted to the highest element type without duplicate nodes. In the image below, a structured surface mesh is converted to a triangular surface mesh. When converting structured zones to unstructured zones, the boundary conditions of the structured zone will be used to define the sub-surfaces of the resulting unstructured zone.


Structured Mesh
Converted Tri-only Mesh

7 Grid Generation Menu

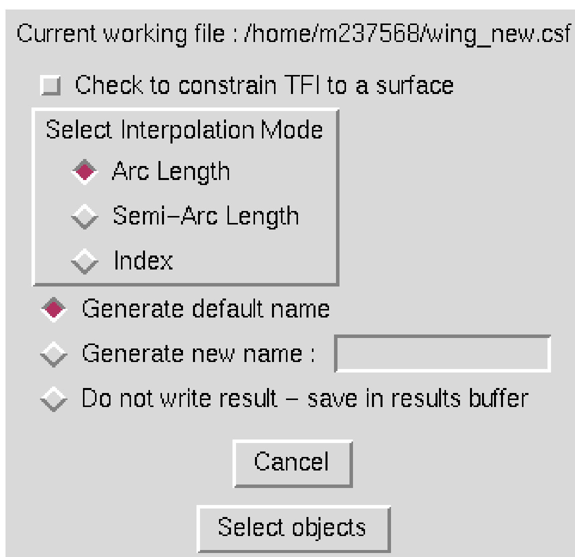
The Modular Aerodynamic Design Computational Analysis Process (MADCAP) “Grid Generation” library and menus are used to redistribute grid points on existing surfaces, generate grids in space between boundary surfaces, or run elliptic smoothing on existing grids.

Selection of the “Grid Generation” menu item produces the drop down menu shown below.

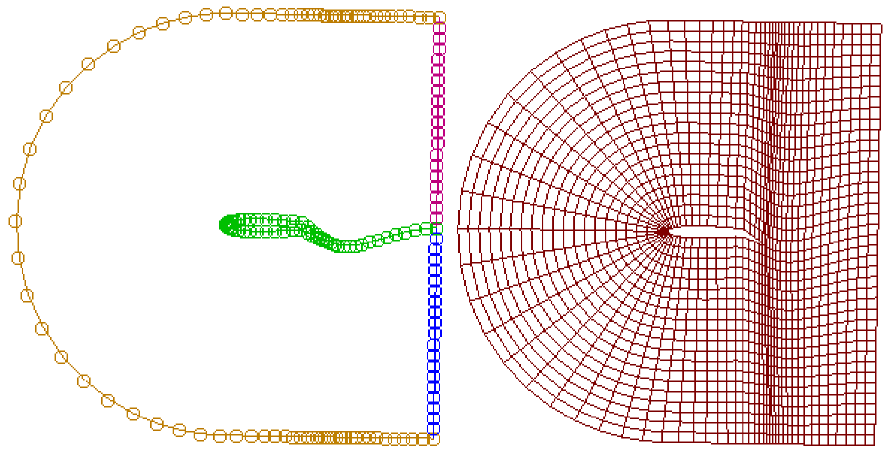


7.1 2D TFI

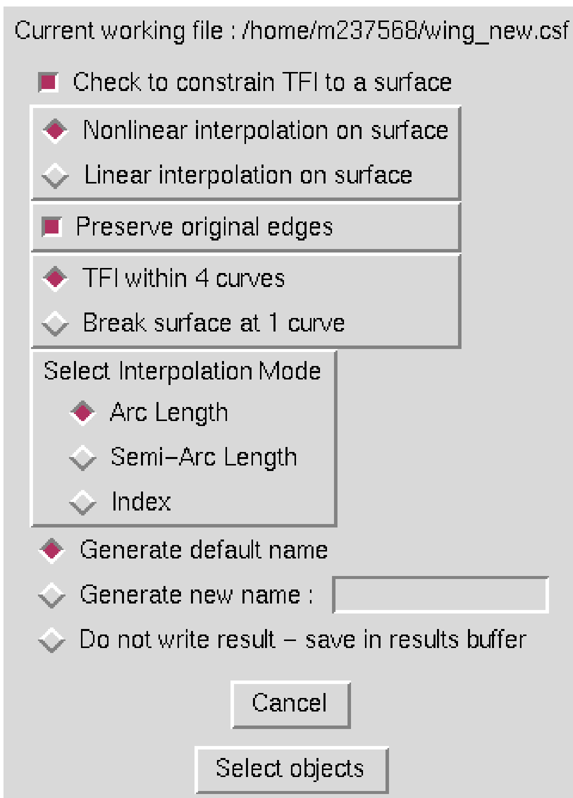
Selection of the “2D TFI ...” submenu causes the following window to be displayed.



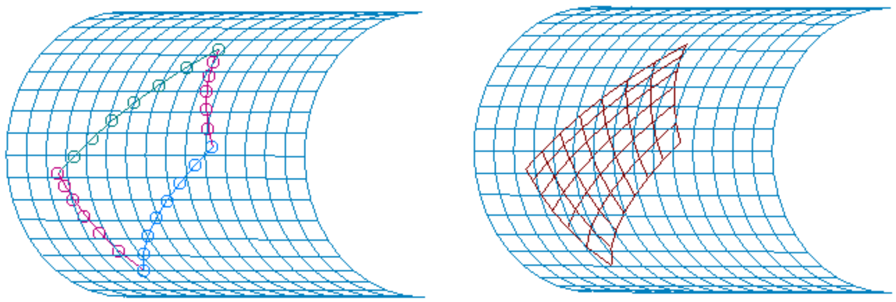
This is the window which is displayed if the checkbutton is not set to constrain the TFI to a structured grid surface. In this case, the user needs only to select the type of interpolation to use, and the program will prompt for four curves to generate a transfinite interpolation within. For this operation, opposite curves must have the same number of points, and the four curves must meet at the endpoints. An example of a surface generated by 2D TFI in space is shown below. The red surface was generated within the four curves. (The surface has been translated to avoid overplotting the four curves.)



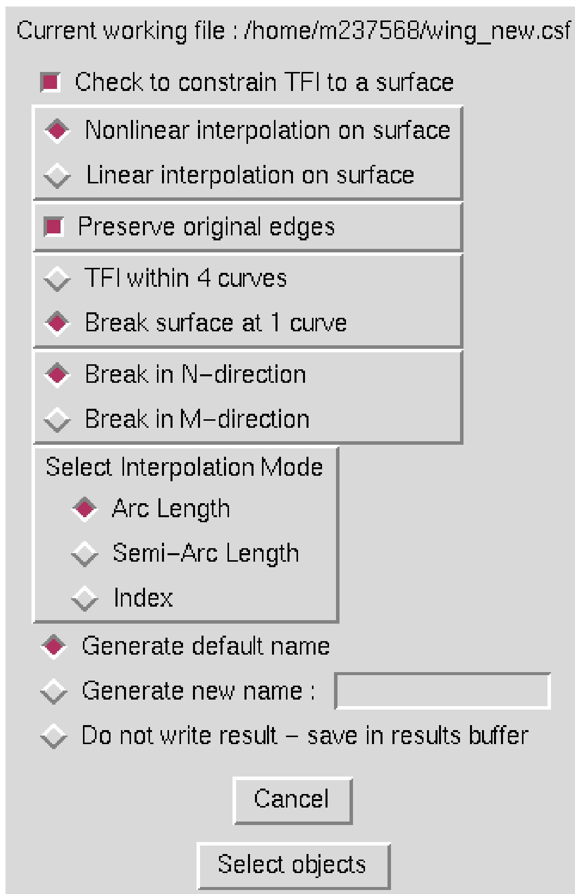
If the box is checked to constrain the TFI to a surface, the window is modified as shown below.



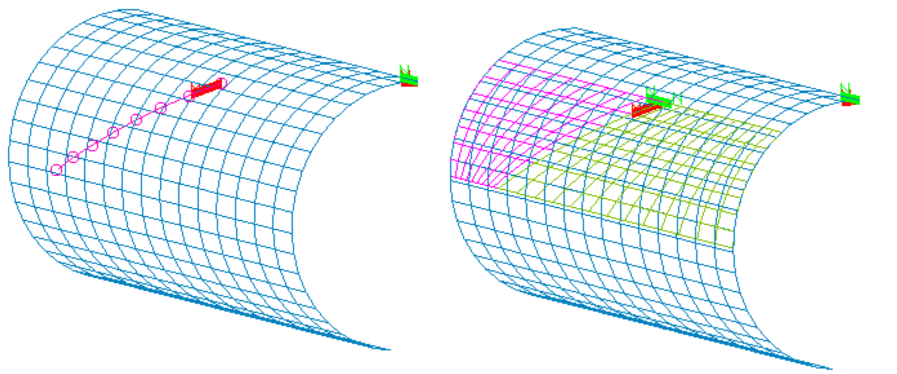
If the option is set to perform TFI within 4 curves, the user must select four curves which lie on the same surface, and the resultant surface will be constrained to lie on the structured grid surface as well. The user must additionally specify an interpolation mode on the structured grid surface (linear or non-linear) and whether to preserve the original edges or use a re-projected computation of the edges. An example of TFI on a structured grid surface is shown in the image below, where the red surface was generated within the four curves on the centerbody.



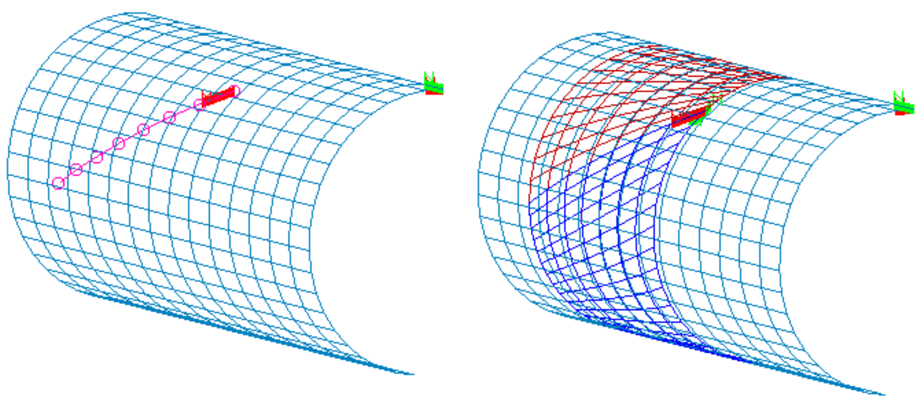
If the radiobutton is set to “Break Surface at 1 Curve”, the window is modified as follows:



In this case, one additional input is required from the user — “Break in N- or M- direction”. For this type of TFI on a surface, only a single curve lying on the surface is required. Two surfaces are generated from the curve, using either the N- or M- location of the endpoints as a constant and interpolating across the other direction to the surface’s extremes. An example helps to clarify this process. In the following example, the surface was broken at the curve in the N-direction. New surfaces are extended out from the curve *in the N direction* to the $N = 1$ and $N = max$ limits of the surface.



In the following image, the surface was broken at the curve in the M-direction. New surfaces are extended out from the curve *in the M direction* to the M = 1 and M = *max* limits of the surface.



In both examples, the generated surface grids have the same number of points as the curve being broken at for the family of lines abutting that curve. The density of the surface grid being broken is used as a guide for generating the grid lines extending to the surface limits.

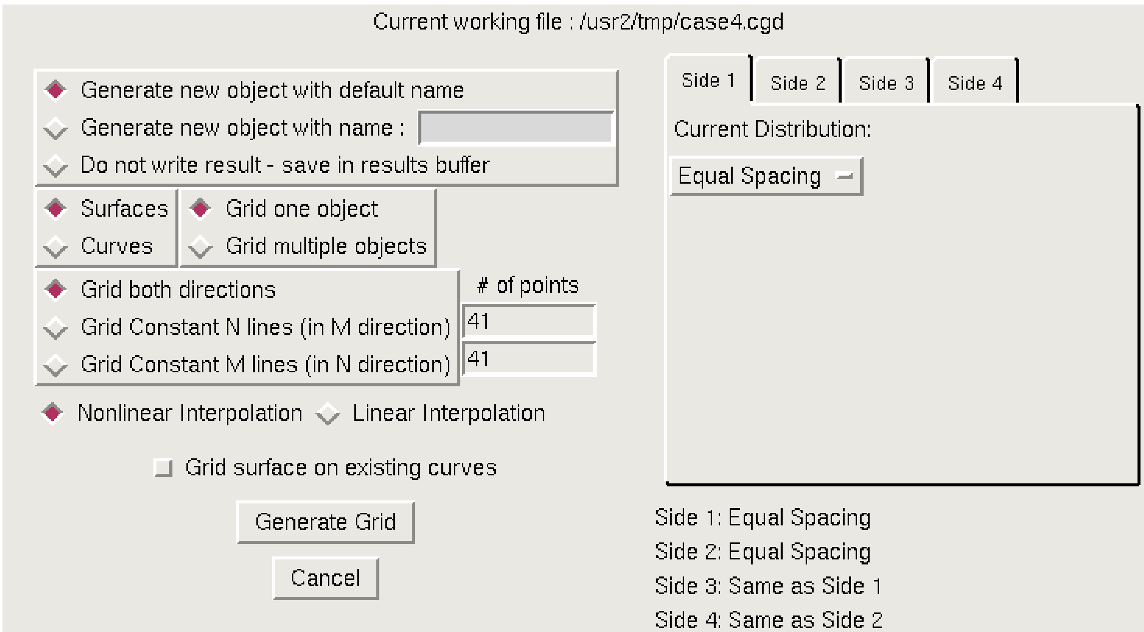
For any type of TFI, the user must also select a radiobutton to indicate whether to generate a default name, use a specified name, or save the result in the results buffer. When the "Select objects" button is selected, the program will prompt for either one or four curves, and for a surface if the TFI is to be constrained to a surface.

The text commands generated by this operation are:

`om select curve` *curve_definition* (One or four instances)
`om select surface` *surface_definition* (If constrained to a surface)
`ggs tfi 2d {space | surface} interp_mode {arc | semi | index} \`
   `[project {linear | nonlinear}] [edges {original | projected}] [dir {n | m}]`
`file save {default | name` *objectname*`}`

7.2 Regrid Surface or Curve

Selection of the “Regrid Surface or Curve ...” submenu causes the following window to be displayed.



This operation is used to change the distribution of grid points on any surface or curve object.

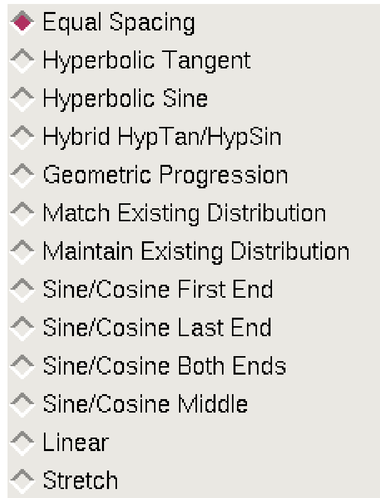
For a surface, to do this in the most general sense the user selects the “Grid both directions” radio button, and specifies a new number of points for each direction (N and M), and a new point distribution function to apply on each of the four sides of the surface. The user can then enter the new number of points desired for both directions and select a distribution function for each of the four sides. The distribution function to be used for each side is determined by first selecting one of the four tabs (Side 1 through Side 4), then selecting the desired distribution from a dropdown menu opened by clicking the entry showing the current distribution. (More on distribution functions later.)

Optionally, the user may choose to only redistribute points on the surface in one direction, leaving the distribution of points in the other direction alone. To do this, the “Grid Constant N lines” or “Grid Constant M lines” radio button should be used, and only the side 1 and 3 distributions (for “Grid Constant N lines”) or side 2 and 4 distributions (for “Grid Constant M lines”) should be specified.

If the “Curves” radio button is used, rather than “Surfaces”, only the Side 1 distribution function should be specified, since only a single distribution function is required for an isolated curve.

Multiple curves or surfaces may be gridded with the same regridding definition by selecting the “Grid multiple objects” radio button. The user must also specify whether to use nonlinear or linear interpolation when redistributing points.

As described above, the distribution function to be used for each side is chosen from a dropdown menu listing the available distributions. The dropdown menu for Side 1 is shown below.



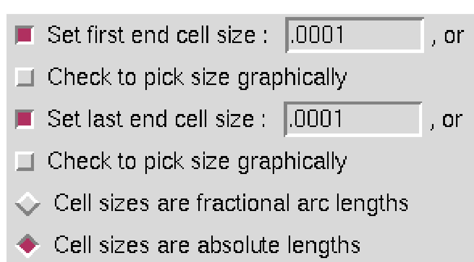
The various distribution functions are described in the following sections.

7.2.1 Equal Spacing

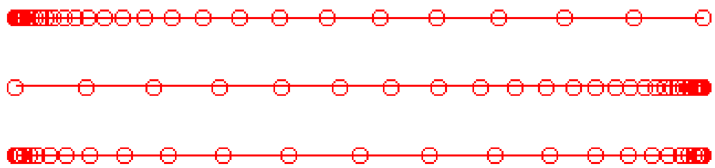
The equal spacing distribution function simply computes the arc length of the curve or surface edge and then distributes the requested number of grid points along the curve such that the distance from one point to the next remains constant. An example line with 31 points equally distributed is shown below.

7.2.2 Hyperbolic Tangent

The hyperbolic tangent distribution function is used to set a fixed cell size at one or both ends of the curve or edge, and distribute the remaining points such that the cell lengths grow or shrink by a hyperbolic tangent function. This function requires several additional inputs as shown in the window below. These buttons appear at the bottom of the list of distribution functions for any side whose option is set to "Hyperbolic Tangent".

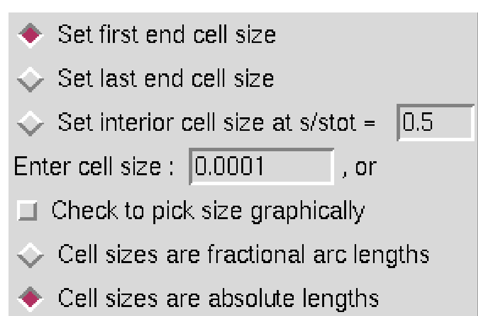


Depending on which checkbuttons are set, the user should enter the desired cell size for each end of the curve. These lengths may also be entered by picking a line segment graphically from the screen. If this mode of defining the cell lengths is chosen, the checkbuttons should be set and the program will prompt for the inputs at the appropriate time. If values are entered in the window, the user should use the final set of radiobuttons to indicate whether the numbers entered should be interpreted as absolute distance values for the length of the first cell or fractional values defining the size of the first cell relative to the total arc length of the curve or edge being regridded. The three curves below show the results of regridding a line with 31 points, with a cell size of .001 of the total arc length, applied at the first end, last end, and at both ends.

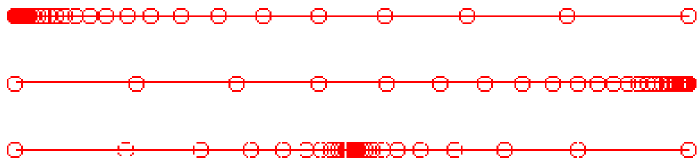


7.2.3 Hyperbolic Sine

The hyperbolic sine distribution function is used to set a fixed cell size at a point on a curve or edge at either end or at an interior location on the curve or edge, and distribute the remaining points such that the cell lengths grow or shrink from this point by a hyperbolic sine function. This function requires several additional inputs as shown in the window below. These buttons appear at the bottom of the list of distribution functions for any side whose option is set to “Hyperbolic Sine”.

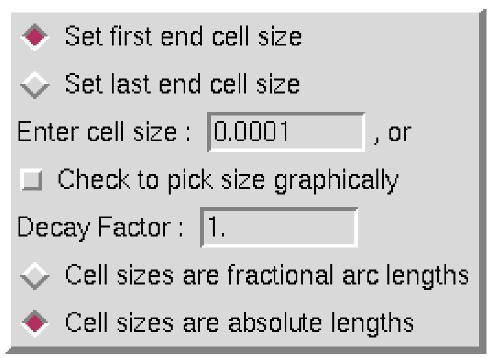


The user should set the desired location for specifying a cell size on the curve. If the “interior cell size” option is used, the user should enter the location on the curve where the specified cell size is to applied, in terms of fractional arc length along the curve. (For example, specifying a location of 0.5 will set the cell size at the center of the curve.) The user also must set the desired cell size at this location. This length may also be defined by picking a line segment graphically from the screen. If this mode of defining the cell length is chosen, the checkbutton should be set and the program will prompt for the input at the appropriate time. If values are entered in the window, the user should use the final set of radiobuttons to indicate whether the numbers entered should be interpreted as absolute distance values for the length of the first cell or fractional values defining the size of the first cell relative to the total arc length of the curve or edge being regridded. The three curves below show the results of regridding a line with 31 points, with a cell size of .001 of the total arc length, applied at the first end, last end, and at a point midway between the two ends.

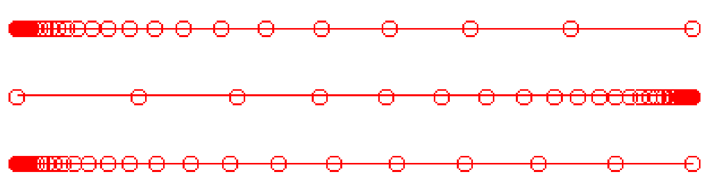


7.2.4 Hybrid HypTan/HypSin

The hybrid hyperbolic tangent/hyperbolic sine distribution function is used to set the cell size at a point on either end of a curve or edge, and distribute the remaining points such that the cell lengths grow or shrink from this point first by a hyperbolic tangent function which blends into a hyperbolic sine function at the opposite end of the curve or edge. This function requires several additional inputs as shown in the window below. These buttons appear at the bottom of the list of distribution functions for any side whose option is set to "Hybrid HypTan/HypSin".

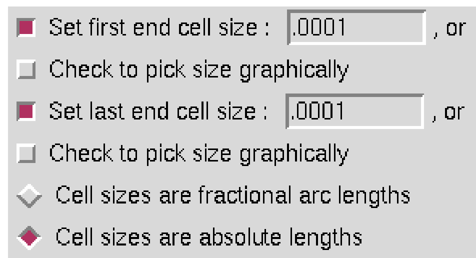


The user should set the desired location for specifying the cell size on the curve, either first end or last end. The user must then set the desired cell size at this location. This length may also be defined by picking a line segment graphically from the screen. If this mode of defining the cell length is chosen, the checkbutton should be set and the program will prompt for the input at the appropriate time. If a value is entered in the window, the user should use the final set of radiobuttons to indicate whether the number entered should be interpreted as an absolute distance value for the length of the cell or a fractional value defining the size of the cell relative to the total arc length of the curve or edge being regridded. The Decay Factor is also required, and determines how quickly the transition from a hyperbolic tangent to a hyperbolic sine function occurs. The three curves below show the results of regridding a line with 31 points, with a cell size of .001 of the total arc length, applied at the first end and last end, with a decay factor of 1. and again at the first end but with a decay factor of 2.

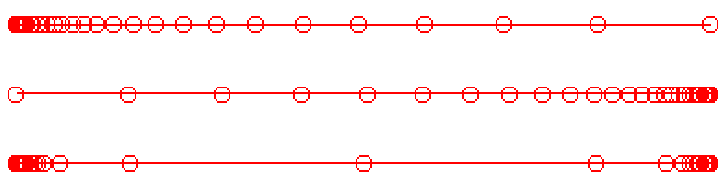


7.2.5 Geometric Progression

The geometric progression distribution function is used to set a fixed cell size at one or both ends of the curve or edge, and distribute the remaining points such that the cell lengths grow or shrink by a geometric progression. This function requires several additional inputs as shown in the window below. These buttons appear at the bottom of the list of distribution functions for any side whose option is set to "Geometric Progression".

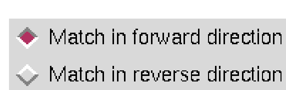


Depending on which checkbuttons are set, the user should enter the desired cell size for each end of the curve. These lengths may also be entered by picking a line segment graphically from the screen. If this mode of defining the cell lengths is chosen, the checkbuttons should be set and the program will prompt for the inputs at the appropriate time. If values are entered in the window, the user should use the final set of radiobuttons to indicate whether the numbers entered should be interpreted as absolute distance values for the length of the first cell or fractional values defining the size of the first cell relative to the total arc length of the curve or edge being regridded. The three curves below show the results of regridding a line with 31 points, with a cell size of .001 of the total arc length, applied at the first end, last end, and at both ends.



7.2.6 Match Existing Distribution

The match existing distribution function is used to regrid a curve or surface edge to match a distribution function on an existing grid line. This is often used when a surface abuts a different surface which has already been regridded to achieve a certain grid distribution. This edge may even be the result of combining different distribution functions by joining curves or surfaces that have been distributed independently. If the matching distribution function is set, the program will prompt the user to graphically select a grid line on the screen to match the distribution on. The following set of radiobuttons is presented at the bottom of the list of distribution functions for any side whose option is set to "Match Existing Distribution".



If the grid line being matched has points running in the opposite direction from the curve being regridded, it may be necessary to specify with these radiobuttons that the distribution function should be reversed before being applied.

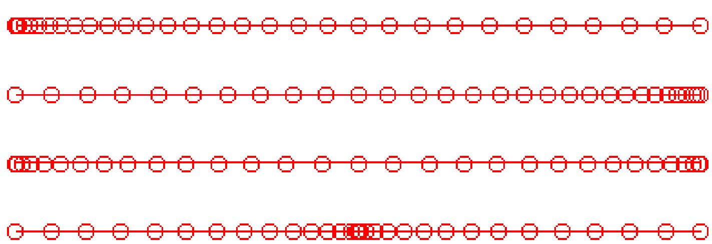
Note that for the Matching distribution function, the user may enter a 0 for the number of points to distribute. In this case, the program will automatically use the same number of points on the curve being regridded as exist on the gridline being matched to. If a non-zero value is entered, the program will compute the distribution function on the curve being matched to, and augment this function to the number of points specified.

7.2.7 Maintain Existing Distribution

The maintain existing distribution function is used to regrid a curve or surface edge using its own current distribution function, but with a new number of points. If the number of points requested is the same as already exist on the edge, an unchanged curve or edge will result. This is often desirable, when changing the distribution on side 1 of a surface but having side 3 remain unchanged. To maintain the current number of points, a 0 may be entered for the number of points when using the "Maintain Existing Distribution" option. If a non-zero value is entered, the program will augment the current function to the number of points specified.

7.2.8 Sine/Cosine Functions

The four sine and cosine distribution functions are used to regrid a curve or surface edge using a pure sine or cosine function with the requested number of points. Depending on the option selected, points may be packed using this function at either the first end, last end, both ends, or in the middle of the curve. The four curves below show the results of distributing 31 points using the sine/cosine functions, at each of the four possible packing locations.

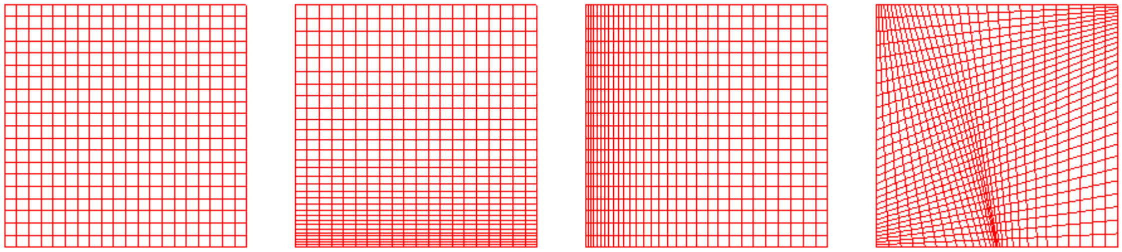


7.2.9 Same as Side 1/2

If the type of object being regridded is a surface, for Sides 3 and 4 the user has the option of simply selecting a "Same as Side 1" or "Same as Side 2" radio button, to apply the same distribution function as specified for the opposite side. This avoids the requirement for respecifying all the regridding inputs when the same function is to be applied all the way across a surface.

7.2.10 Surface Regrid Examples

The image at the upper left below shows a simple square with 21 points in an equal arc distribution on each of the four sides. The additional images show the results of regridding this surface with 31 points, with varying distribution functions on the four sides. Examples include regridding only the grid lines in either of the directions, and regridding all four edges differently.



The text commands generated by the Regrid function are:

```
om select {curve | surface} object_definition
ggs set_dist {side1 | side2 | side3 | side4} diststring
```

where *diststring* is one of

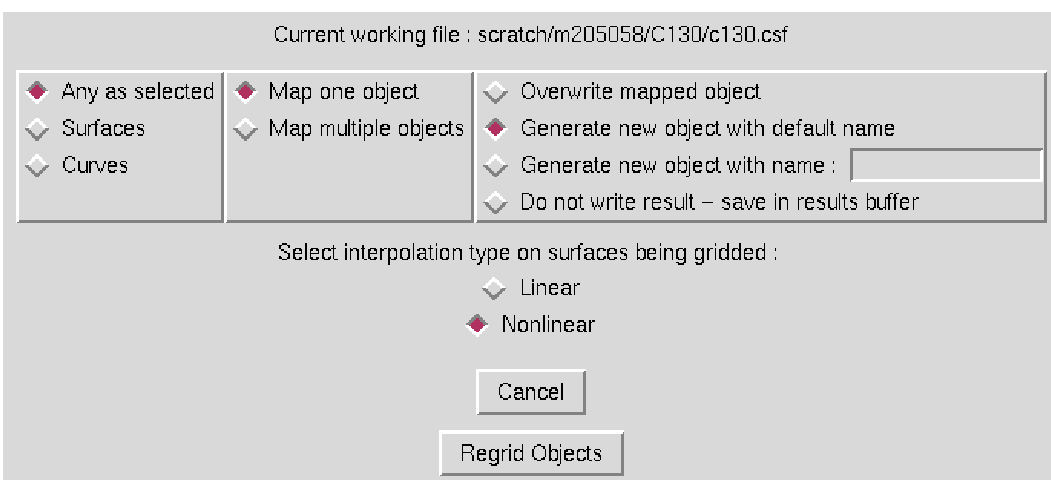
```
hyptan {[first s] [last s]} [method {absolute | relative}]
geom {[first s] [last s]} [method {absolute | relative}]
hypsin {[first s | last s | interior s loc s]} [method {absolute | relative}]
hybrid {[first s | last s]} [method {absolute | relative}] decay d
sincos {first | last | both | interior}
matching {forward | reverse}
{equal | nochange | opposite}
```

and finally

```
ggs redist dir {n | m | both} [interp {linear | nonlinear}] [n n] [m m]
file save {default | name surfname}
```

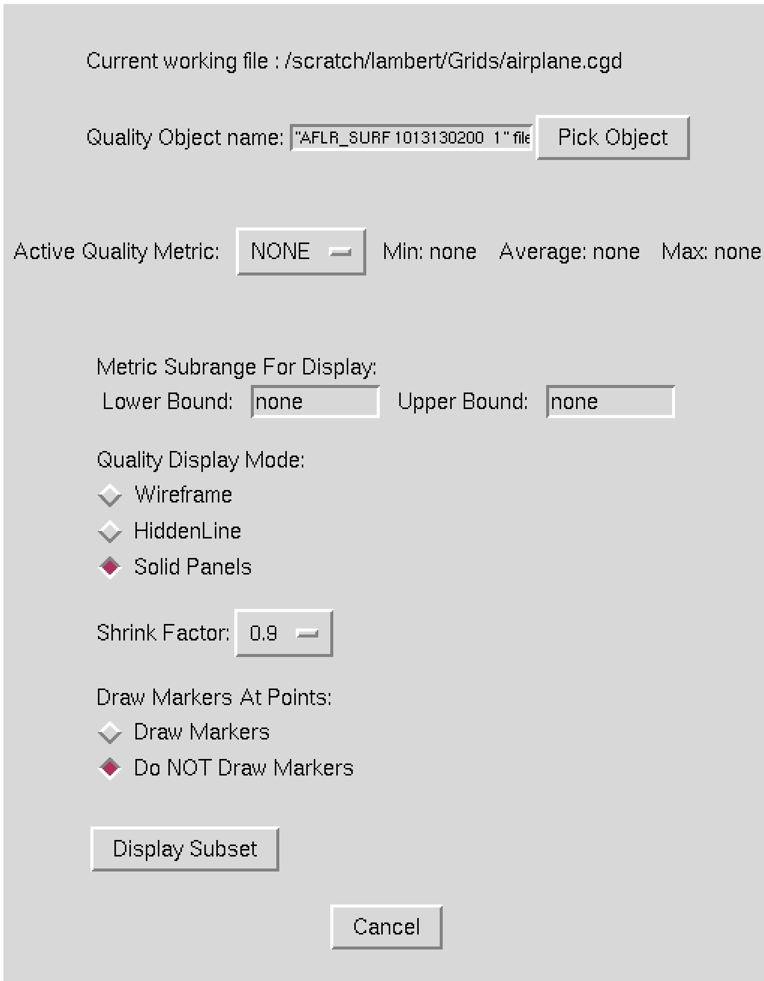
7.3 Map Surface or Curve onto Another

Selection of the "Map Surface or Curve onto Another ..." submenu causes the following window to be displayed.



7.4 Unstructured Mesh Quality

Selection of the "Unstructured Mesh Quality ..." submenu causes the user to be prompted to select an unstructured object in the graphics window. After selecting the object, the following window is displayed.



In this form, the user specifies the criteria for computing and displaying the quality of an unstructured mesh. Mesh quality on unstructured surface and volume meshes can be computed and displayed. The "Active Quality Metric:" drop down box is used to specify which quality metric should be computed. Depending on the type of object selected, surface and cell quality metrics may appear in this list. The only surface quality metric that is available is the "Minimum Edge Ratio". This metric is a measure of the ratio of the shortest edge over the longest edge in a face. Once a quality metric is selected, the minimum, maximum and average values for that metric will be listed beside the "Active Quality Metric" drop down box.

The user can now specify the options for display of the metric information. A subrange of the cells or faces to display is input in the "Metric Subrange For Display:" text entry boxes. Here the user can specify a lower and upper bound of the metric. Any faces/cells that fall within the specified range will be displayed when the "Display Subset" button is picked.

Example: For a case where the “Face Edge Ratio” quality metric provides a minimum value of 0.1, a maximum of 0.8 and an average of 0.7, specification of an “Upper Bound” of 0.2 will display all faces with a edge ratio between 0.1 and 0.2. Note that in this example, “none” is left in the “Lower Bound” text entry box to signify that no lower bound should be applied.

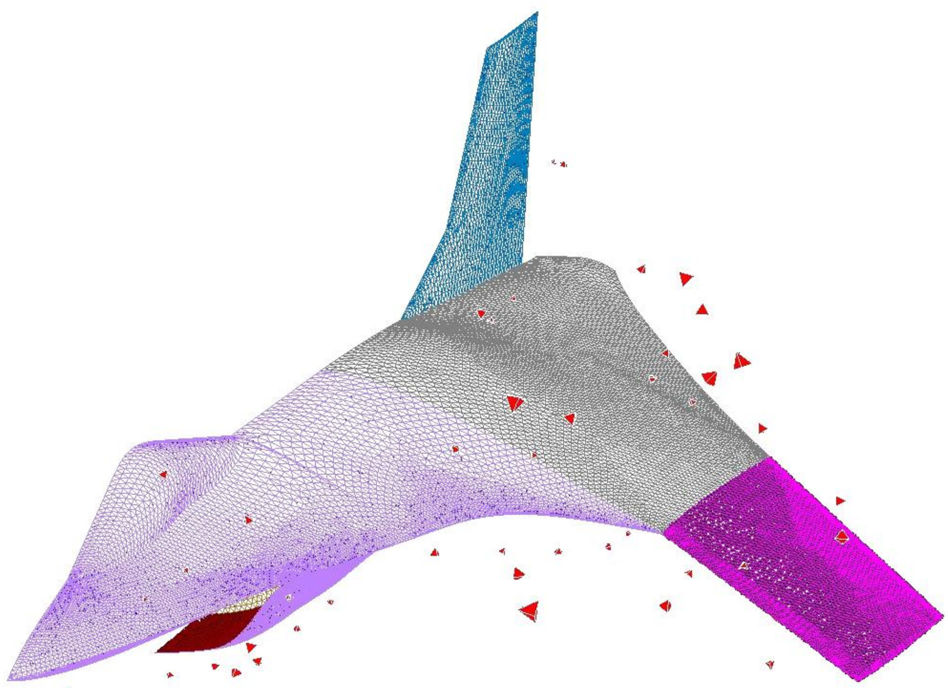
The “Quality Display Mode:” radio button boxes specify whether to draw the selected faces/cells in “Wireframe”, “HiddenLine”, or “Solid Panels” display mode.

The “Shrink Factor:” drop down box allows the user to specify a percentage to shrink the face/cell before displaying it. This option is useful for distinguishing individual cells for faces that touch each other.

The “Draw Markers At Points:” radio button specifies whether markers should be drawn at the nodes that make up each selected face or cell. This option is useful for highlighting the poor cells or faces in cases where they may otherwise not be visible. An example is the display of a triangular face that has collapsed to a degenerate edge. In this case, the face would appear as a single line and may be difficult to see or locate. By drawing the markers at the nodes, the face will be easier to see.

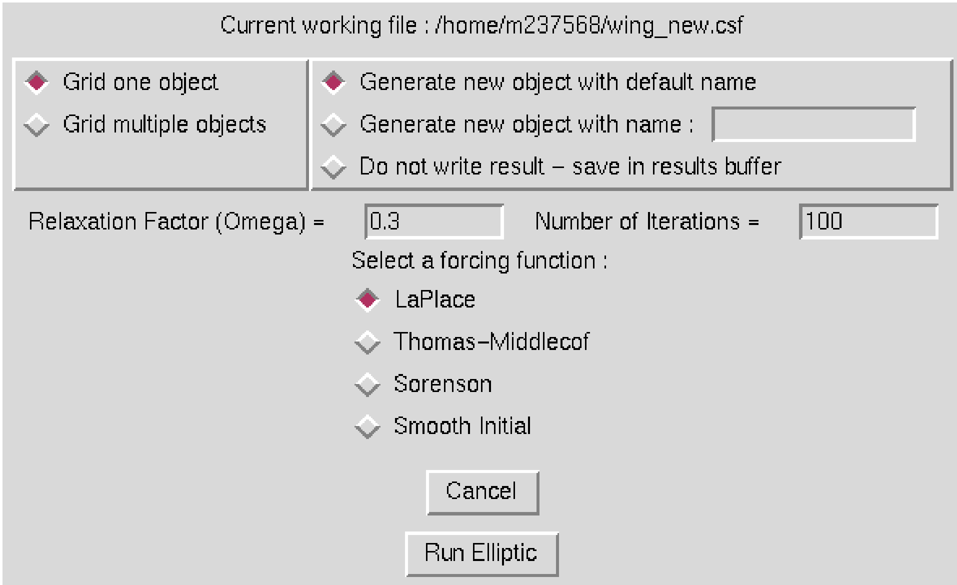
Once the metric has been selected and the display options specified, the “Display Subset” button will display the selected faces/cells. The options may be adjusted and the display updated by reselecting the “Display Subset” button at any time.

An example quality display for a cell quality metric is shown below.

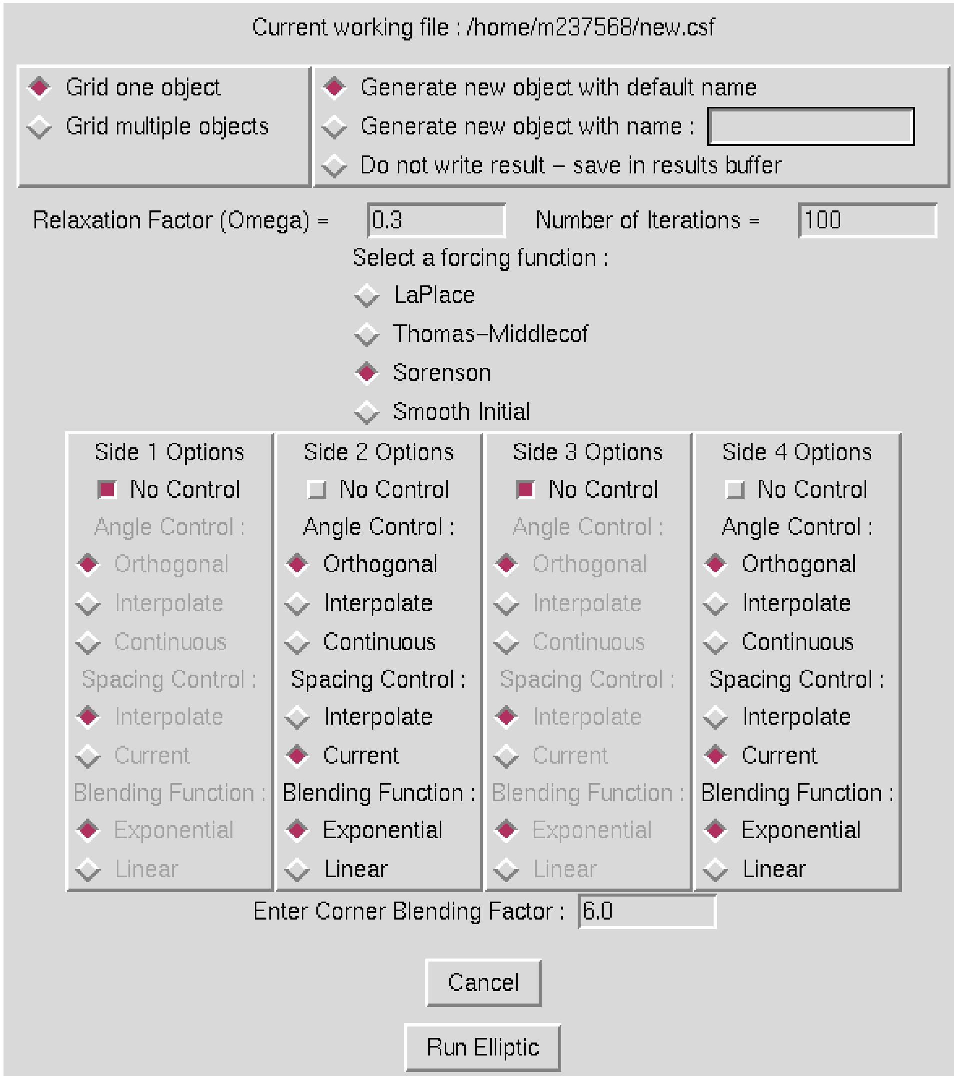


7.5 2D Surface Elliptic

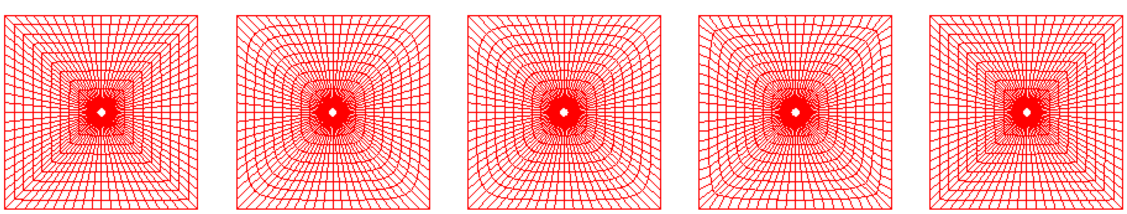
Selection of the “2D Surface Elliptic ...” submenu causes the following window to be displayed.



This operation is used to use elliptic smoothing to smooth the grid lines on a surface. Four methods are available for elliptic smoothing. LaPlace and Thomas-Middlecof smoothing require no further inputs beyong a relaxation factor (omega) and the number of iterations to run, both of which are entered in the entry boxes on the window. For the "Smooth Initial" function, additional inputs of an N- and M-subiteration count is also required (entry boxes will appear on the screen for these inputs). The most robust of the functions is the Sorenson. In this case, the window is modified as shown below.



Under Sorenson elliptic smoothing, the user can control the angle grid lines approach edges, how the spacing off the wall is controlled from one end of an edge to the other, and how the options on different edges are blended near corners. These options can be independently specified on each of the four edges of a surface. A single corner blending factor is can also be set in the entry box at the bottom of the screen. The four images to the right below show the result of running elliptic smoothing on the typical polar grid inside a square region shown on the left. In order, the images are the results for LaPlace, Thomas Middlecof, Sorenson, and Smooth Initial elliptic options running 30 iterations with the default options.



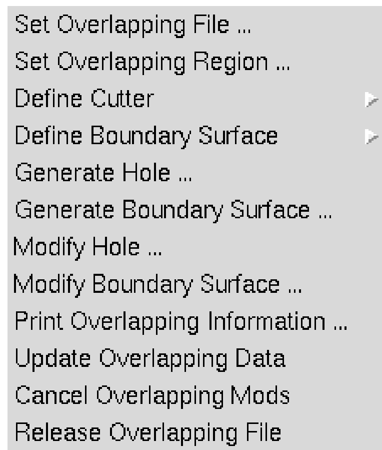
When elliptic smoothing is run on a surface grid, the surface may be defined linearly or nonlinearly for interpolation purposes. After the "Run Elliptic" button is selected, the user will be prompted for the surface to smooth, and a new surface will be generated. This object should be specified as having a default name, a specified name, or to be saved in the results buffer. The text commands generated may include any of the following.

```
om select surface surface_definition
elp set relaxation factor omega
elp set forcing function 2d {laplace | middlecof | sorenson | smooth}
elp set sorenson siden control {none | \
   angle {orthogonal | interpolate | continuous} \
   spacing {interpolate | current} blending_function {exponential | linear}}
elp set sorenson blending_factor factor
elp set smooth iterations {[mdir num] [ndir num]}
elp elliptic [function {laplace | middlecof | sorenson | smooth}] \
   [interp {linear | nonlinear}] iterations iters
file save {default | name surfname}
```

7.6 Surface/Curve Smoothing

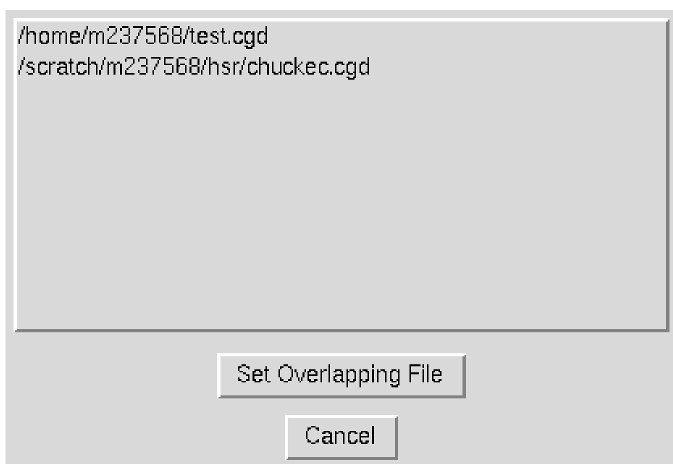
8 Overlapping Grid Menu

Selection of the Modular Aerodynamic Design Computational Analysis Process (MADCAP) “Overlapping” menu item produces the drop down menu shown below.



8.1 Set Overlapping File

Selection of the “Set Overlapping File ...” submenu causes the following window to be displayed.



The current list of *.cgd* files that have been opened in MADCAP is displayed in the list box in this window. (See Section 3.2 for details on opening files in MADCAP.) Since overlapping hole and fringe data (IBLANK data) is stored only in the *.cgd* file, other file types currently open in MADCAP are not displayed in this list.

Selecting a file name from this list will cause it to be highlighted. When the desired file has been selected, pick the “Set Overlapping File” button at the bottom of the window. The window will be shut down and a message will appear in the MADCAP message box indicating that the file has been set as the Overlapping file. Only one file can be set as the Overlapping File at any given time.

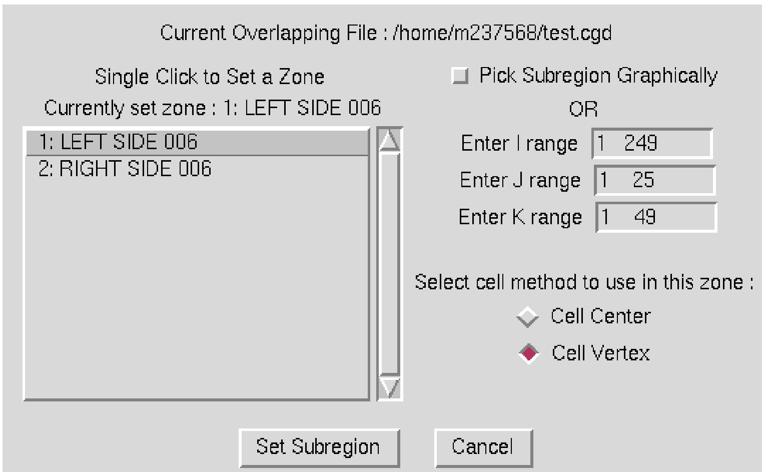
To escape from this window with no action being taken, select the "Cancel" button.

The associated text command is:

`ol file name` *filename*

8.2 Set Overlapping Region

Selection of the "Set Overlapping Region ..." submenu causes the following window to be displayed.



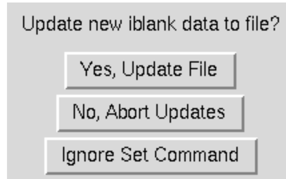
This window is used to set the zone and subregion of that zone in which the overlapping operations are to be applied. A zone and subregion must be set before most of the overlapping operations can be performed. The scrollable list on the left of the window contains the list of zones in the currently set overlapping file. (See Section 8.1). A single click on a zone name in this list will change the currently set zone to the selected zone. The name of the currently set zone is displayed at the top of this list, and is defaulted to the first zone in the list the first time the window is displayed.

Immediately to the right of the scrollbar are a set of entry boxes in which to enter I, J, and K subranges. By default, the ranges reflect the complete zone. The user can modify these ranges to any subregion of the full zone that is desired. If the user would prefer to select the corners of the subregion graphically instead, the checkbutton marked "Pick Subregion Graphically" should be highlighted. This will disable entries in the boxes, and the user will be prompted to pick the corners of the subarea from the graphics screen after exiting the window.

Finally, two radiobuttons are provided in which the user should indicate whether the zones specified are to be operated on using a cell-centered or cell-vertex method.

After ensuring that the correct zone and region have been set, select the "Set Subregion" button at the bottom of the screen. This will cause the window to close, and a message will appear in the graphics region indicating that the desired area has been set. If the subregion was specified to be picked graphically, the program will prompt the user to pick the corners for the subregion, using the left mouse button to pick points from the displayed surfaces.

If the user attempts to change the overlapping region or file after changes have been made but before selecting the "Update Overlapping Data" submenu, the following window is presented.



Selecting the "Yes, Update File" button has the same effect as selecting the "Update Overlapping Data" submenu (Section 8.10). Selecting the "No, Abort Updates" button has the same effect as selecting the "Cancel Overlapping Mods" submenu (Section 8.11). Selecting the "Ignore Set Command" button ignores the attempt to change the region or file, with no changes being made to the current settings.

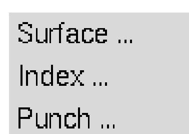
To escape from the Set Overlapping Region window with no action being taken, select the "Cancel" button.

Exiting this window causes the following text commands to be issued.

`ol zone` *zone number*
`ol set {cell center | cell vertex}`
`ol subregion {I` *range* `J` *range* `K` *range* `| all}`

8.3 Define Cutter

Selection of the "Define Cutter" submenu causes the following cascade menu to be displayed.

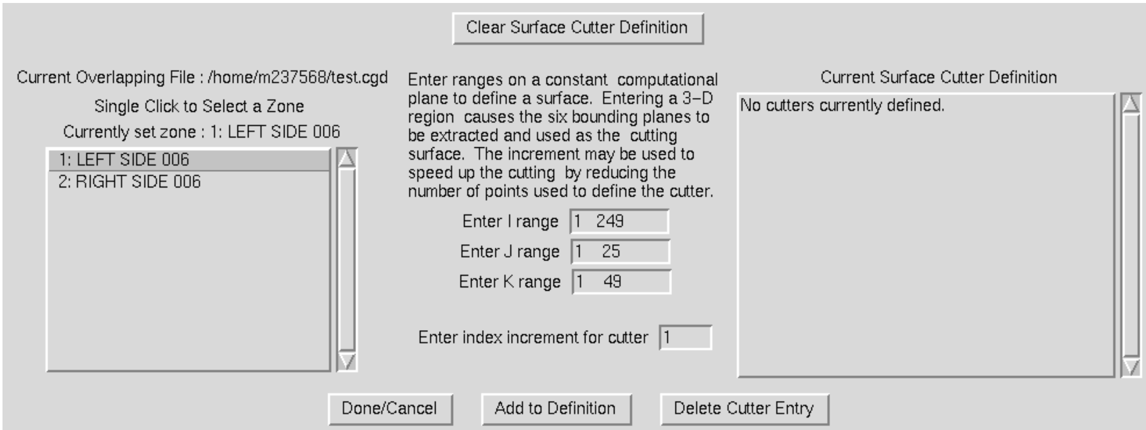


Three types of cutters may be used to cut holes in the selected zone and subregion. Each type requires its own unique inputs. Selecting one of the types from the cascade menu will cause a window to be displayed for entering these inputs. The three types of cutters are:

- Surface
- Index
- Punch

8.3.1 Define Surface Cutters

Selecting the "Surface ..." submenu item causes the following window to be displayed.

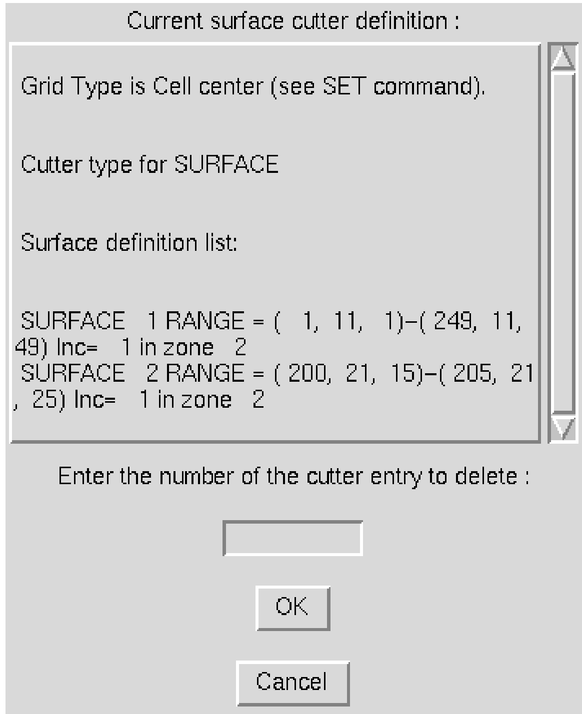


A surface cutter is used to define a cutting surface by computational planes in any zone. In the classic example of a wing zone overlapped on a background grid, the user may want to set a cutting surface at the J = 21 plane of the wing grid and cut away all points in the background zone falling inside this surface. To define a surface cutter in this window, the user first selects a zone from the list of zones in the box at the left of the window. This causes the full zone ranges to appear in the I, J, and K range entry boxes at the middle of the window.

To define a range of points on a computational surface of the zone, the user should modify the entry boxes to identify a constant I, J, or K plane and a subregion of the other two directions. If a three-dimensional region is entered, the cutting surface will be defined by the set of six bounding planes making up the region.

The entry box for an "index increment" can be used to speed up the hole generation operation. The default value is 1, meaning that every point in the computational range entered is used to define the cutting surface. By entering larger integer values, a coarser definition of the cutting surface can be described which may speed up the hole generation.

After defining the correct surface cutter, the user should select the "Add to Definition" Button at the bottom of the window. This will cause the defined surface cutter to be added to the current surface cutter definition. The current definition is shown in the box to the right of the window. After the definition has been added, the window is redrawn with the entered data added to the "Current Surface Cutter Definition" box. If no further surface cutters need to be defined, the "Done/Cancel" button should be selected. If the user wants to delete one of the components in the current definition, the "Delete Cutter Entry" button should be selected, and the following window will be displayed.



The box at the top is the same cutter definition as in the previous window, but the user now has an entry box in which to enter the numeric identifier for the component of the surface cutter to be deleted. Enter the number in the entry box and hit the "OK" button to cause the specified part of the surface cutter to be deleted.

To completely clear out the current surface cutter definition, the "Clear Surface Cutter Definition" button at the top of the main definition window should be selected.

Note that this window is used only to *define* a cutter, and does not cause any holes to be cut itself. After all cutters that are desired have been defined, the "Generate Hole" menu (Section 8.5) should be used to actually proceed with hole cutting.

Use of this window causes a form of the following command to be issued.

`ol cutter surface [new | {range | region} [in] zone` *zonnum range* `[increment` *n*`]]`

If the Delete Cutter window is used, the following command is issued.

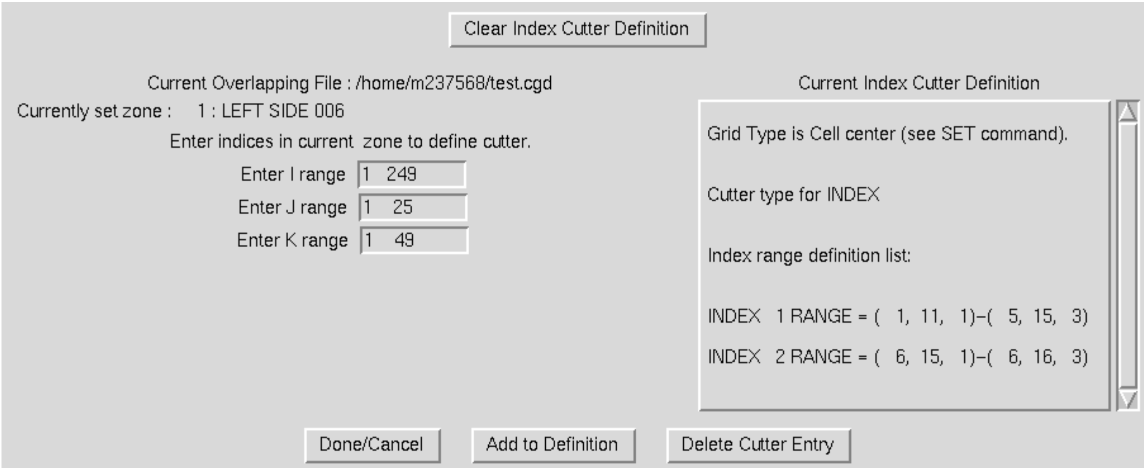
`ol modify cutter delete` *entrynum*

If the Clear Cutter button is used, the following command is issued.

`ol cutter surface clear`

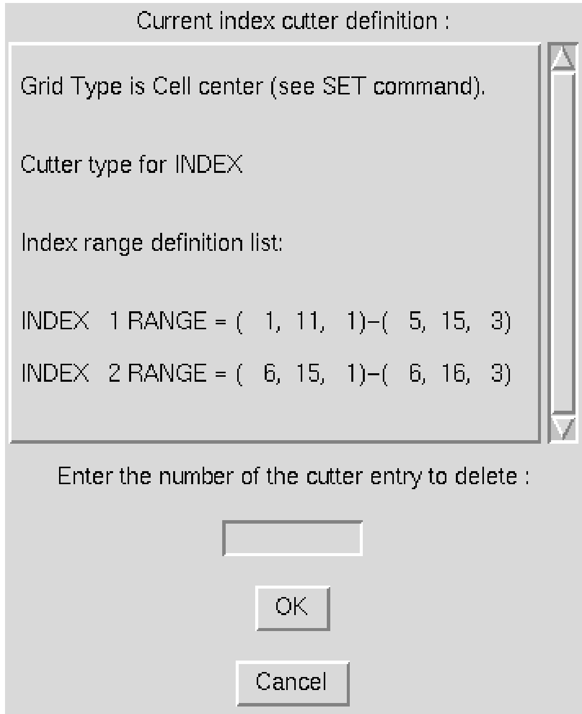
8.3.2 Define Index Cutters

Selecting the "Index ..." submenu item causes the following window to be displayed.



An index cutter is used to define a set of points in the current zone that should be marked as hole points. All points falling within the specified range will be marked to be set as hole points when the "Generate Hole" command (Section 8.5) is issued. The full zone dimensions are initially entered in the I, J, and K range entry boxes in this window. To set a different range of points, the user simply enters the desired values in the entry boxes.

After setting the correct range of points, the user should select the "Add to Definition" Button at the bottom of the window. This will cause the defined zone range to be added to the current index cutter definition. The current definition is shown in the box to the right of the window. After the definition has been added, the window is redrawn with the entered data added to the "Current Index Cutter Definition" box. If no further index ranges need to be specified, the "Done/Cancel" button should be selected. If the user wants to delete one of the components in the current definition, the "Delete Cutter Entry" button should be selected, and the following window will be displayed.



The box at the top is the same cutter definition as in the previous window, but the user now has an entry box in which to enter the numeric identifier for the component of the index cutter to be deleted. Enter the number in the entry box and hit the "OK" button to cause the specified part of the index cutter to be deleted.

To completely clear out the current index cutter definition, the "Clear Index Cutter Definition" button at the top of the main definition window should be selected.

Note that this window is used only to *define* hole regions, and does not actually change the points in the file . After all ranges that are desired have been defined, the "Generate Hole" menu (Section 8.5) should be used to actually proceed with setting the points to hole points in the file.

Use of this window causes a form of the following command to be issued.

`ol cutter index [range]` *range*

If the Delete Cutter window is used, the following command is issued.

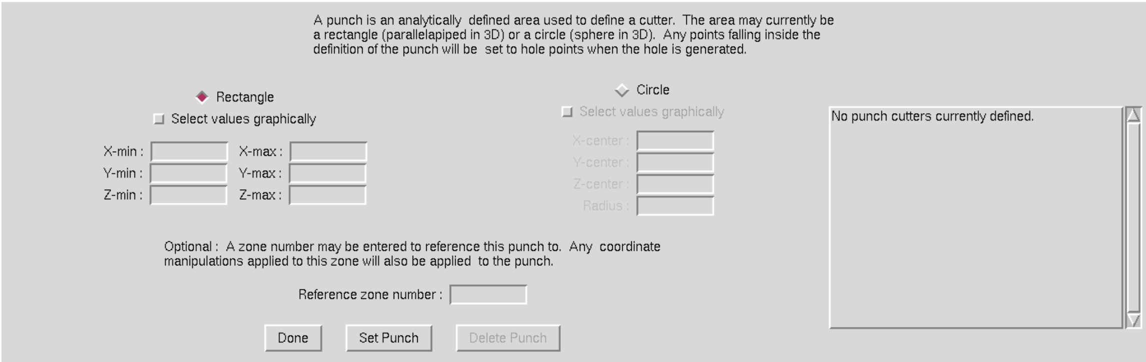
`ol modify cutter delete` *entrynum*

If the Clear Cutter button is used, the following command is issued.

`ol cutter index clear`

8.3.3 Define Punch Cutters

Selecting the "Punch ..." submenu item causes the following window to be displayed.



A punch is an analytically defined surface which can be used to indentify points to be set to hole points, similar to a paper hole punch being used to create a circular hole in a sheet of paper. Two types of punches can currently be defined in MADCAP — a rectangular box or a sphere. Two radiobuttons are provided to set the type of punch to be defined, with the rectangular punch the initial default. For a rectangular punch, the user enters the x, y, and z coordinates of two opposite corners of the rectangular region in the entry boxes. The entry boxes for the circular punch are disabled when the Rectangle radiobutton is selected. When the Circle radiobutton is selected, the entry boxes for the x, y, and z coordinates of the center of the sphere and the radius of the sphere are enabled, and the Rectangle entry boxes are disabled. For both the rectangle min/max and the sphere center/radius, the user has the option of highlighting the "Select values graphically" checkbutton. In this case, the entry boxes may remain empty, and the program prompts the user to identify these values via screen picks from the displayed surfaces.

For some multidisciplinary applications where overlapping grids are automatically moved, like store separation, it may be desirable to use a punch to cut the initial hole. As the store moves and rotates against the fixed background grid, it is necessary that the punch used to define the hole move and rotate with the grid. For this purpose, an optional Reference zone number may be entered in the bottom entry box. This "anchors" the punch to a specific zone in the file, such that the punch definition is updated along with any motion of the specified zone.

After the punch has been defined, the user should select the "OK" button to set the punch definition. To escape without defining a punch, select the "Cancel" button. Note that unlike the surface cutter (Section 8.3.1) and index cutter (Section 8.3.2) only one punch definition can be defined at a time. Specifying a new punch definition will replace the current punch cutter definition. If more than one punch needs to be defined, the user should define a punch, generate its hole using the Generate Hole command described in Section 8.5, and then define the next punch.

Use of this window causes a form of the following command to be issued.

`ol cutter punch [{rectangle | circle}` *coord_data* `[REF_ZONE` *zonnum*`]]`

where *coord_data* has the form:

`min` *xval yval zval* `max` *xval yval zval* (For a rectangular punch)
`center` *xval yval zval* `radius` *val* (For a circular punch)

8.4 Define Boundary Surface

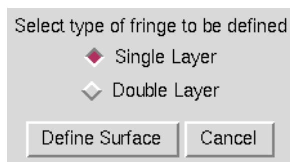
Selection of the "Define Boundary Surface" submenu causes the following cascade menu to be displayed.

Three types of boundary surfaces may be defined in the selected zone and subregion. Each type requires its own unique inputs. Selecting one of the types from the cascade menu will cause a window to be displayed for entering these inputs. The three types of boundary surfaces are:

- Fringe
- Face
- Index

8.4.1 Define Fringe Boundary Surface

Selecting the "Fringe ..." submenu item causes the following window to be displayed.



A fringe boundary surface is used to mark the points surrounding a hole as a boundary on which a boundary condition can be set (a "fringe"). The only input required for setting this type of boundary surface is to specify whether a single layer or double layer of points surrounding a hole are to be marked as fringe points. Two radiobuttons are provided for this purpose. After selecting the desired option, the user should select the "Define Surface" button to set a boundary fringe definition. To escape with no action being taken, select the "Cancel" button.

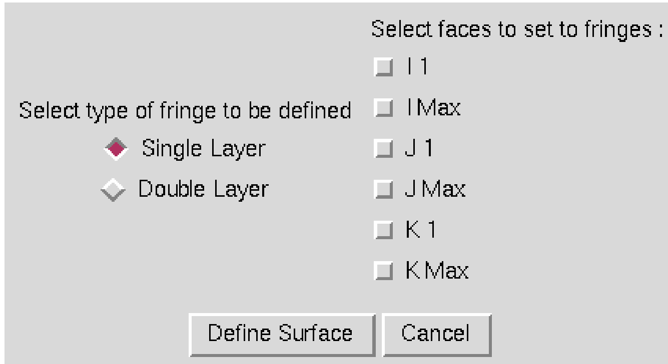
Note that this operation does not actually set the fringe points, but only the mode to use when the Generate Surface command (Section 8.6) is used.

Using this window causes the following commands to be issued.

```
ol srfacer [mode] {single | double}
ol srfacer [mode] fringe
```

8.4.2 Define Zone Face Boundary Surface

Selecting the "Face ..." submenu item causes the following window to be displayed.



This mode is used when all of the points on one of the six computational faces of a zone are desired to be set to fringe points. As with the basic fringe boundary surface around a hole (Section 8.4.1), a full face fringe boundary surface may be generated as a single layer or double layer of points. The user selects the appropriate radiobutton for this option at the left of the window, and then uses the checkbuttons to the right of the window to identify any set of faces to be marked as fringe boundaries. After setting the desired options, the "Define Surface" button should be selected. To escape with no action being taken, the "Cancel" button is selected.

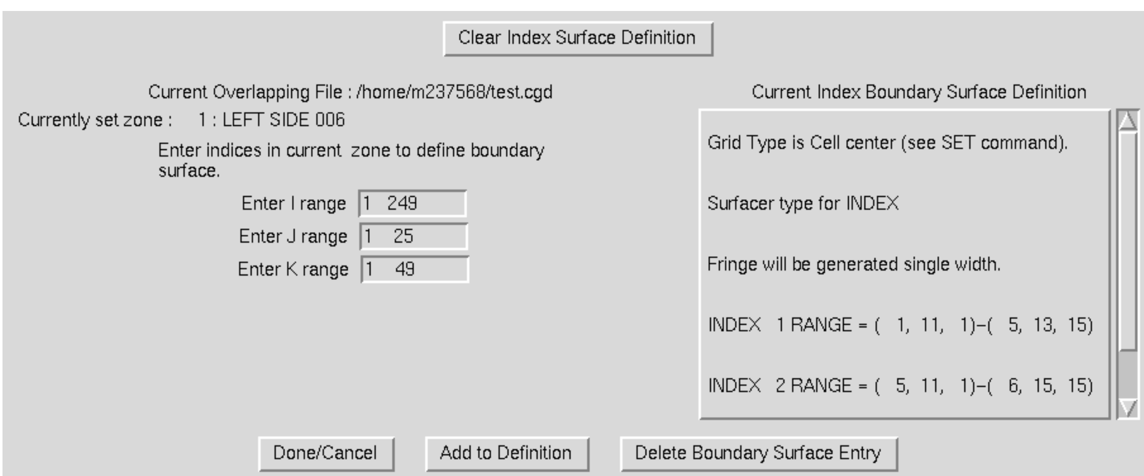
Note that this operation does not actually set the fringe points, but only the mode to use when the Generate Surface command is used, as described in Section 8.6.

Using this window causes the following commands to be issued.

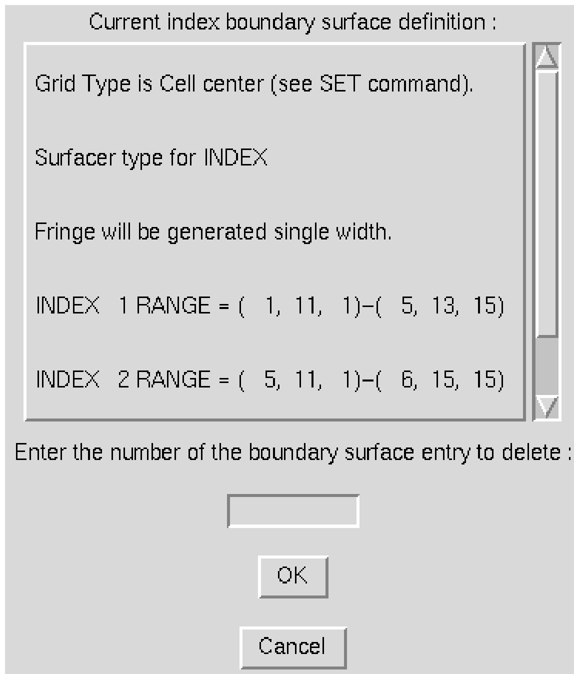
```
ol srfacer [mode] {single | double}
ol srfacer [mode] face [[on | off] [i1] [imax] [j1] [jmax] [k1] [kmax] ...]
```

8.4.3 Define Boundary Surface by Index

Selecting the "Index ..." submenu item causes the following window to be displayed.



The index boundary surface is used to specify a set of points in the current zone which are to be defined as a boundary surface on which a boundary condition can be set. All points within the ranges of points specified will be able to be assigned a boundary surface id which can then have a boundary condition specified. The entry boxes for I, J, and K ranges to the left of the window are initially set to the full zone. The user can modify these entry boxes to set the desired region of the zone. After setting a range, the “Add to Definition” box should be selected to add the specified range to the current index boundary surface definition. The window will then be redrawn with the specified ranges added to the current definition seen in the box to the right of the window. If the user wants to delete a certain entry from the current boundary surface definition, the “Delete Boundary Surface Entry” button should be used and the following window will be displayed.



The box at the top is the same surface definition as in the previous window, but the user now has an entry box in which to enter the numeric identifier for the component of the index surface to be deleted. Enter the number in the entry box and hit the “OK” button to cause the specified part of the index boundary surface to be deleted.

To completely clear out the current index surface definition, the “Clear Index Surface Definition” button at the top of the main definition window should be selected.

Note that this window is used only to *define* points to be set as boundaries, and does not actually change the points in the file. After all ranges that are desired have been defined, the “Generate Boundary Surface” option described in Section 8.6 should be used to actually proceed with setting the points to boundary points in the file.

Use of this window causes a form of the following command to be issued.

`ol srfacer [mode] index [range]` *range*

If the Delete Surface Entry window is used, the following command is issued.

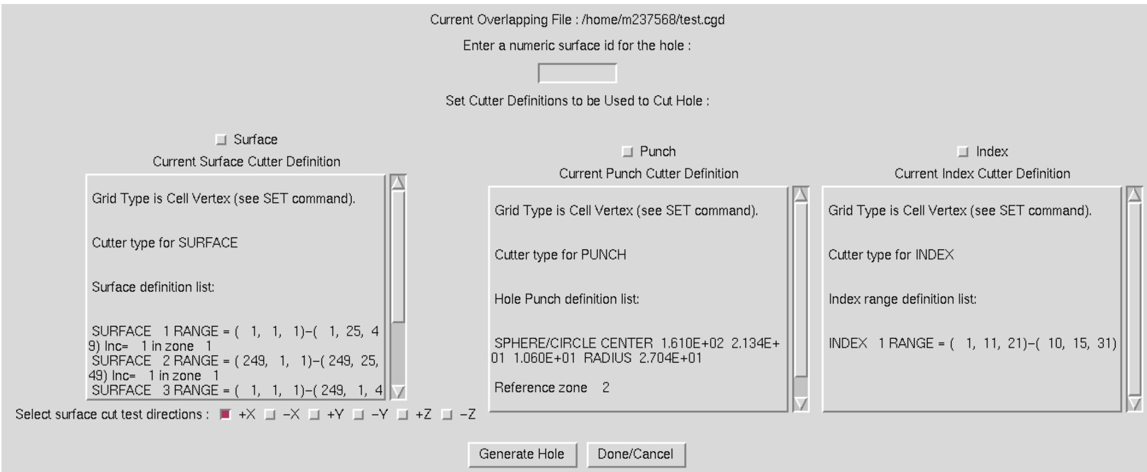
`ol modify srfacer delete` *entrynum*

If the Clear Index Surface button is used, the following command is issued.

`ol srfacer [mode] index clear`

8.5 Generate Hole

Selection of the "Generate Hole" submenu causes the following window to be displayed.



This window is composed primarily of scrollable text boxes containing the currently set definitions of the three types of cutters — surfaces (Section 8.3.1), punches (Section 8.3.3), and index ranges (Section 8.3.2). At the top of the window is an entry box in which the user should set a numeric surface id to be used to label the hole generated by this command. After setting the id, the user should select any or all of the checkbuttons at the top of the text boxes to indicate which cutter definitions should be used to generate holes with the input surface id. Additionally, for surface cutter definitions, the user should specify which directions to use for the test to determine if a point falls inside or outside the defined surface cutter. Checkbuttons are provided at the bottom of this textbox for this purpose.

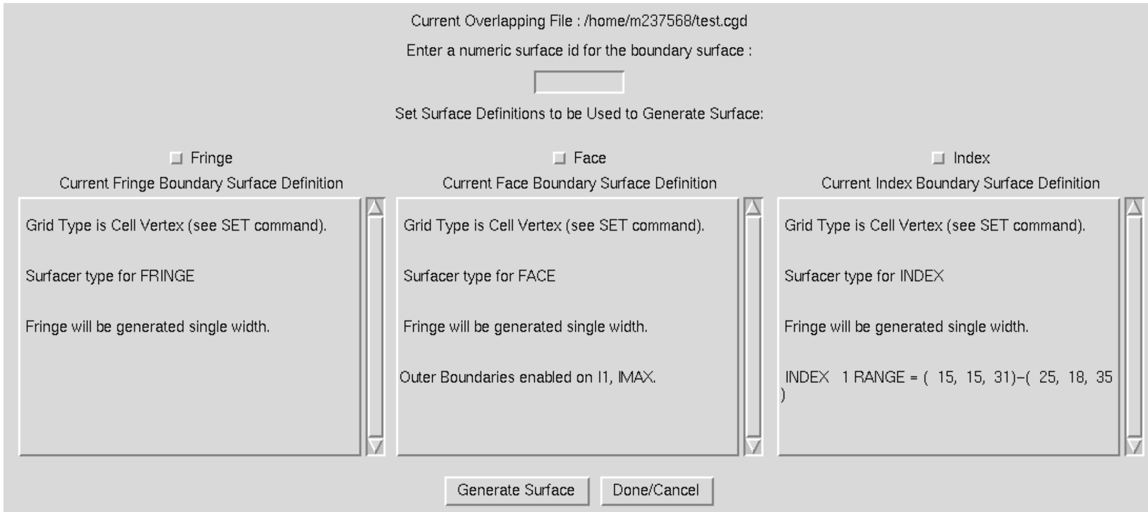
Holes of different types may be specified to be cut at the same time with this command, but all will have the same numeric id after they have been generated. When the desired data has been entered, the "Generate Hole" button should be selected and MADCAP will proceed to cut the holes. Informational messages are output to the message box, and the graphic display is updated to reflect the new surface iblank data (depending on the setting of the blanking mode in the main viewing menu, as described in Section 4.12). To escape this window with no action being taken, select the "Done/Cancel" button.

Use of this window may cause any of the following commands to be issued.

```
ol surface [surface_name | all]
ol set cut test directions [[[on | off] {+/-x | +/-y | +/-z | all}] ...]
ol cutter {surface | punch | index}
ol generate hole
```

8.6 Generate Boundary Surface

Selection of the "Generate Boundary Surface" submenu causes the following window to be displayed.



This window is composed primarily of scrollable text boxes containing the currently set definitions of the three types of boundary surfaces — hole fringes (Section 8.4.1), zone faces (Section 8.4.2), and index ranges (Section 8.4.3). Checkbuttons at the top of these boxes are used to set the types of boundary surfaces to be generated with this command. At the very top of the window is an entry box in which the user should set a numeric surface id to be used to label the fringe to be generated. If the "fringe" boundary surface type is selected, the fringe will be generated around the hole with the same label as the one entered. For the other types, the entered surface id will be used for the new fringe generated.

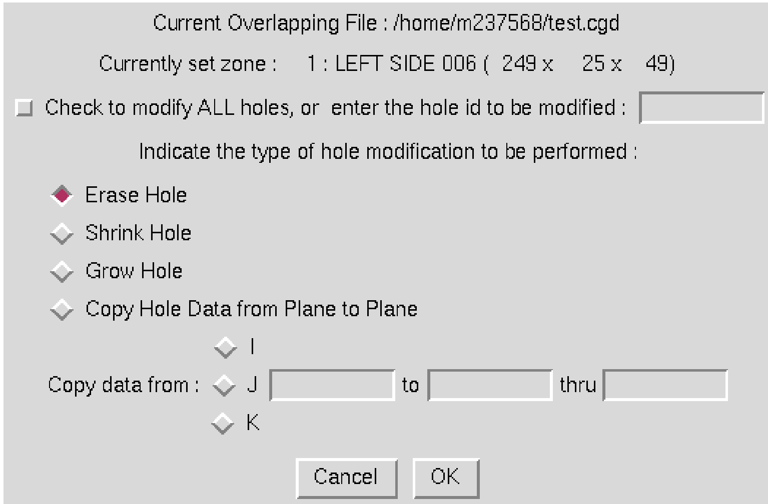
When the correct data has been entered, the "Generate Surface" button should be selected. To exit with no action being taken, select the "Done/Cancel" button.

Use of this menu will cause the following commands to be issued.

```
ol surface [surface_name | all]
ol srfacer {face | fringe | index}
ol generate surface
```

8.7 Modify Hole

Selection of the "Modify Hole" submenu causes the following window to be displayed.



Hole modification can be performed on either all existing holes in the defined subregion (Section 8.2), or on a specific hole. To modify all holes, set the checkbutton at the top of the window. Otherwise, enter the id for the hole to be modified in the entry box. Four types of hole modification can be performed. First, holes can be erased completely. Note that this also erases any associated fringes that may have been generated around the hole. A hole can also be shrunk or grown using the next two radiobuttons. In these cases, an extra layer of points are either removed or added to the hole. Finally, with the "Copy" radiobutton, the hole points with the specified id that are currently set on a given I, J, or K plane can be copied to a different plane or range of planes. If the "copy" radiobutton is selected, one of the I, J, or K radiobuttons, and the entry boxes for the plane to copy from and the plane range to copy to must be filled in.

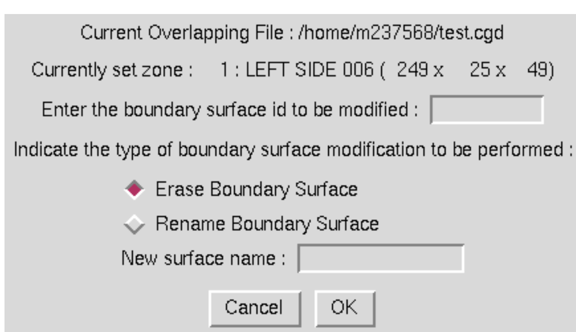
After the correct options have been set, the user should select the "OK" button. To escape with no action being taken, the "Cancel" button should be selected. To save the modifications to file, the update main menu option (Section 8.10) must still be selected.

Use of this menu will cause the following commands to be issued.

```
ol surface [surface_name | all]
ol modify hole {erase | shrink | grow | copy {i | j | k m to n1 [thru n2]}}
```

8.8 Modify Boundary Surface

Selection of the "Modify Boundary Surface" submenu causes the following window to be displayed.



Boundary surface modification can take two forms — either erasing the stored definition of a boundary surface, or renaming a boundary surface. In either case, the surface id to be modified must be entered in the entry box at the top of the window, and the appropriate radiobutton should be selected. If the "Rename" option is selected, the new name should be entered in the box provided.

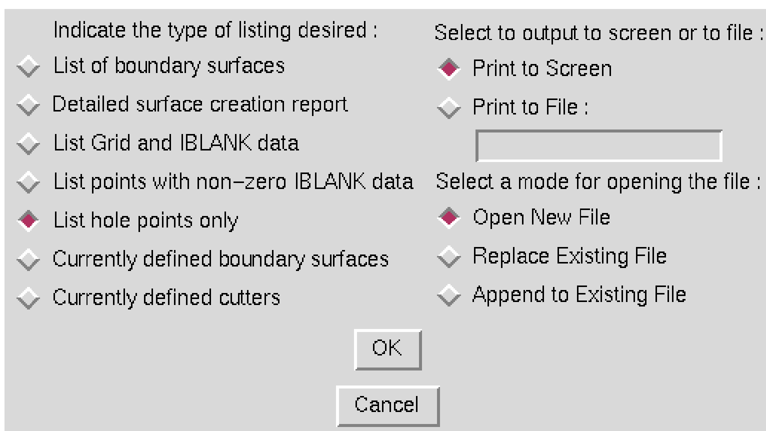
After the correct options have been set, the user should select the "OK" button. To escape with no action being taken, the "Cancel" button should be selected. To save the modifications to file, the update main menu option (Section 8.10) must still be selected.

Use of this menu will cause the following commands to be issued.

`ol surface [`*surface_name*` | all]`
`ol modify surface {erase | rename from `*old_name*` to `*new_name*`}`

8.9 Print Overlapping Information

Selection of the "Print Overlapping Information" submenu causes the following window to be displayed.



This window is used to set the type of overlapping information to be reported on, and where to send the desired output. Seven types of listing can be obtained — a list of the holes and boundary surfaces in the current subregion, a detailed report on how holes and surfaces were created, a listing of the grid (x, y, z) and iblank data in the current region, a listing of only the points with iblank data set, a listing of hole the hole points, the currently defined boundary surfaces, and the currently defined hole cutters. The type of listing obtained is controlled by the radiobuttons on the left. On the right, the user may choose to print the listing either to the screen or to file. If the Print to File radiobutton is selected, the user must enter a file name for the listing and whether to open a new file, overwrite an existing file, or append to an existing file.

The associated text command is:

```
ol print {surface list | surface creation report | grid | iblank holes | \
   srfacer mode | cutter mode}
```

If output is directed to a file, MADCAP precedes the boundary condition print command with a file open command as follows:

`file open list `*filename*` [mode {new | replace | append}]`

8.10 Update Overlapping Data

Selection of the "Update Overlapping Data" submenu causes the overlapping data generated in the currently set overlapping region (Section 8.2) to be written to the currently set overlapping file (Section 8.1). Until this action has been taken, the overlapping data is only updated in program memory, and not to the actual *.cgd* file.

8.11 Cancel Overlapping Mods

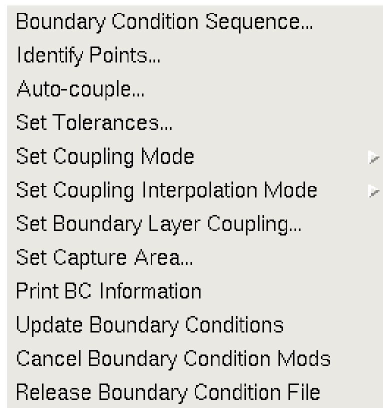
Selection of the "Cancel Overlapping Mods" submenu causes the overlapping data that has been set on the currently set overlapping region (Section 8.2) to be canceled out, and replaced with the original set of overlapping data from the currently set overlapping file (Section 8.1). This action is non-recoverable, as all overlapping information stored for the current region in program memory is restored to the file state existing before the set overlapping data operations were begun. To save the changes to file, use the "Update Overlapping Data" submenu described in Section 8.10.

8.12 Release Overlapping File

Certain regions of program memory are allocated when an overlapping file is first set, and used throughout the overlapping process. This memory remains allocated and usable as long as an overlapping file is set. If the MADCAP user has completed all desired overlapping operations, it may be desirable to use the "Release Overlapping File" submenu to free up the memory associated with setting overlapping data. This is never a required operation, but deallocates resources that may be required elsewhere. After selecting this submenu, no overlapping file will be set, and the user must respecify a file before again using the overlapping menus.

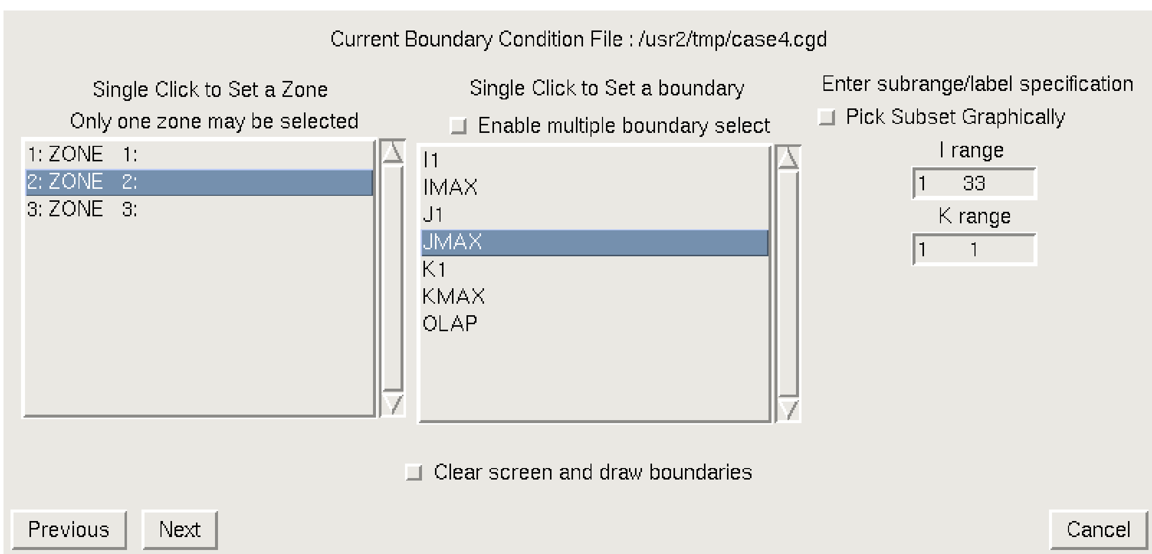
9 Boundary Condition Menu

Selection of the Modular Aerodynamic Design Computational Analysis Process (MADCAP) "Boundary Conditions" menu item produces the drop down menu shown below.



9.1 Boundary Condition Sequence

Selection of the "Boundary Condition Sequence ..." submenu causes the following window to be displayed.



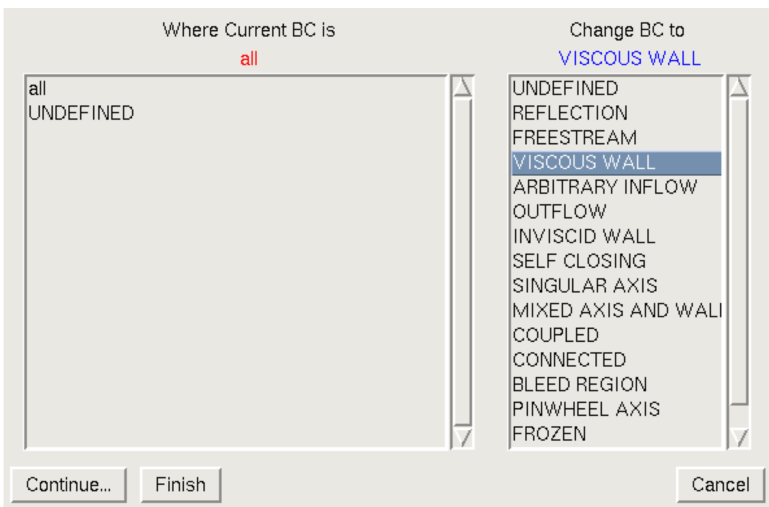
This window is used to set the zone, and regions of that zone, at which a unique boundary condition is to be set. The scrollable list on the left of the window contains the list of zones in the current working file. A single click on a zone name in this list will change the currently set zone to the selected zone, and display a list of boundaries in that zone in the box to the right.

When a single boundary from the boundary list is selected, the applicable index ranges are displayed to the right. These allow the user to select index subranges in the case of a structured mesh, or a point range for unstructured surfaces, at which the boundary condition should be applied. The subrange of the selected computational boundaries is, by default, the entire boundary. If a smaller subregion of a boundary is desired, the limits may be entered in the entry boxes provided. Alternatively, the user may turn on the "Pick Subset Graphically" checkbox, and the user will be prompted to pick the corners of the subarea from the graphics screen after exiting the window.

Multiple boundaries may be selected simultaneously by first picking the radio button labeled "Enable multiple boundary select" at the top of the boundary list, then selecting the desired boundaries while holding down the Control key. This allows the same boundary condition to be quickly set on more than one boundary. Note that this only applies when a boundary condition is being set for the entire boundary; index subranges cannot be set when multiple boundaries are selected.

If the checkbox at the bottom of the screen is enabled, MADCAP will clear the screen and redraw only the boundaries that have been set in this window. This is often desirable, especially if you are setting the subset graphically, since it will cause the selected face to be redrawn and rescaled at the center of the screen where it is easy to pick from.

After ensuring that the correct zone and regions have been set, select the "Next" button at the bottom of the screen to close the current window and open the "Assign/Change Boundary Condition" window.

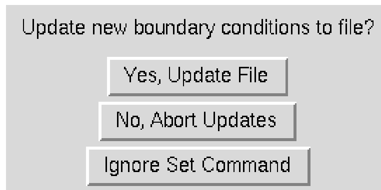


To the left of this window is a list of all boundary conditions currently set for the specified zone and region, as well as the special keyword "all". The user may change the boundary conditions of all points in the region by choosing the "all" keyword from the list, or change boundary conditions selectively by choosing any of the existing boundary condition types to affect only points with that boundary condition. The list box on the right contains the full list of boundary conditions that are available.

When the desired change types and new boundary conditions have been chosen, selecting "Finish" at the bottom of the window will close the window. Note that at this point the boundary conditions are updated *in program memory only*, not in the *.cgd* file itself. To actually update the boundary conditions in the *.cgd* file, select "Update Boundary Conditions" from the main Boundary Conditions menu. (See Section 9.10.) You can also select "Cancel Boundary Condition Mods" to delete the boundary condition update, leaving the boundary conditions in the *.cgd* file unchanged. (See Section 9.11.)

If additional boundary conditions are to be set, instead of selecting “Finish” at the bottom of the “Assign/Change Boundary Condition” window, you can select “Continue” to go back to the “Set Boundary Condition Zone and Boundary” window. Then select the desired zone for the additional boundary conditions.

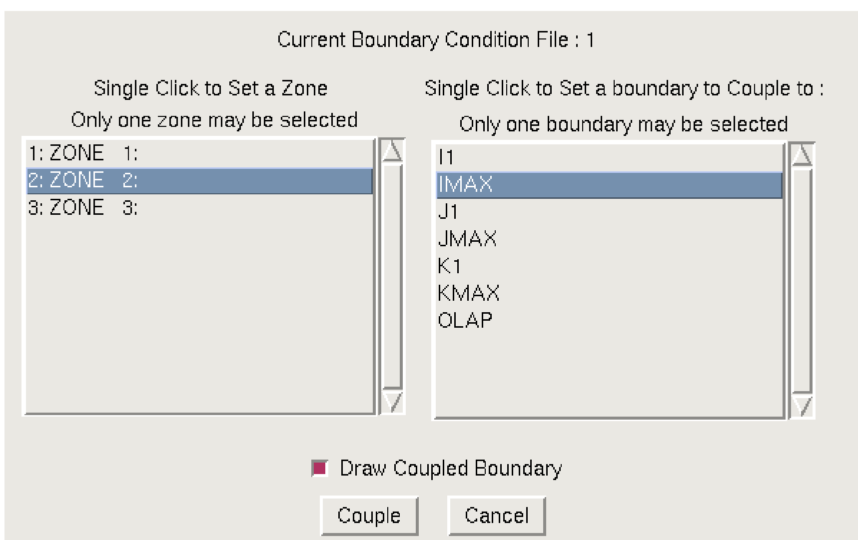
At this point, since the boundary conditions that were chosen for the previously-selected zone and region haven’t yet been updated in the *.cgd* file, the following confirmation subwindow will pop up.



Selecting the “Yes, Update File” button has the same effect as selecting “Update Boundary Conditions” from the main Boundary Conditions menu, described in Section 9.10. Selecting the “No, Abort Updates” button has the same effect as selecting “Cancel Boundary Condition Mods” from the main Boundary Conditions menu, described in Section 9.11. Selecting the “Ignore Set Command” button ignores the attempt to set a new zone for additional boundary conditions.

After finishing with the confirmation subwindow, you can continue the procedure described above to set additional boundary conditions.

Several of the possible boundary condition specfications cause additional input windows to be displayed. If the “Coupled” boundary condition is specified in the “Assign/Change Boundary Condition” window, the following window is displayed.

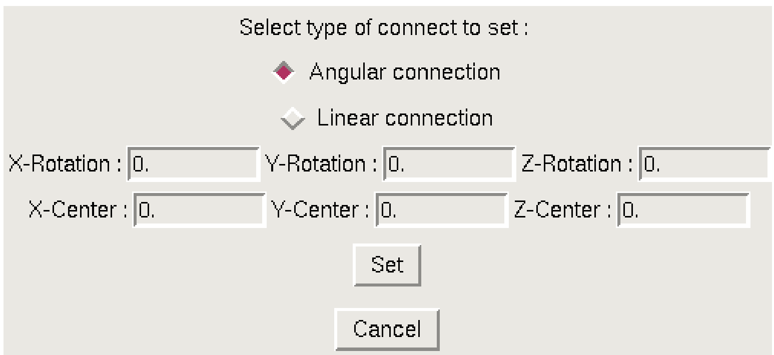


This window is used to set the zone and boundary that the currently-specified boundary condition region is to be coupled to. If the current region consists of zone faces, the OLAP boundary must not be chosen as the boundary to couple to, or an error will be generated. The OLAP boundary can only be used when the subregion consists of overlapping labels. The zone face buttons can be used for any type of coupling. The list box on the left shows the list of zones in the current boundary condition file. Selecting a zone from the list on the left, and then a computational face (or OLAP) from the list on the right, will set them as the zone and boundary to which the currently-specified boundary condition region will be coupled.

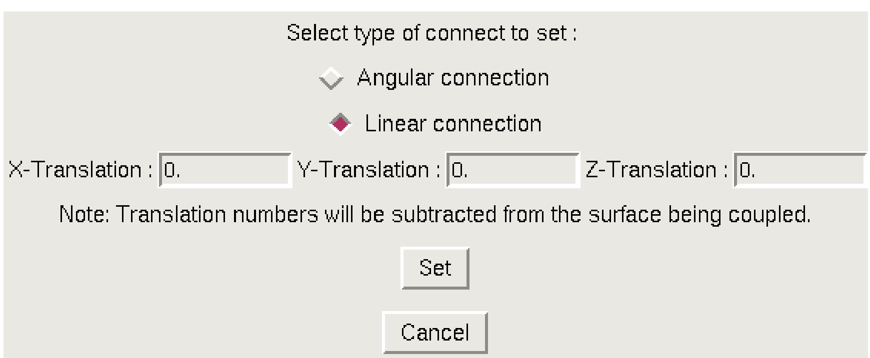
The checkbutton at the bottom of the window is used to force MADCAP to draw the selected coupling boundary before actually proceeding with the coupling. This may be useful to help visualize whether the correct set of points has been coupled.

When the correct zone and boundary have been set, the "Couple" button is selected to proceed with the zone coupling. To escape from this window with no action being taken, select the "Cancel" button.

If the "Connected" boundary condition is specified in the "Assign/Change Boundary Condition" window, the following window is displayed.

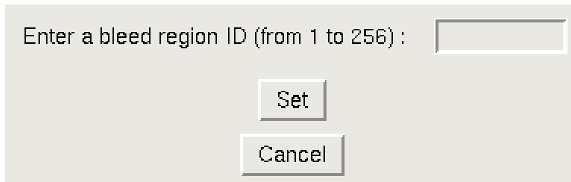


This window is used to set both the type of connection to be performed, as well as the required information for that type of connection. At the top of the window are two radiobuttons, used to specify whether an angular or linear type of connection is to be performed. The window is displayed above as it appears for angular connection, in which case the window also displays six entry boxes in which to set a center of rotation for the angular connection, and the amount of rotation about each axis to be performed to accomplish the connection. If the radiobutton is switched to the linear connection type, the window is modified as shown below.



In this case, only three entry boxes are available, in which to specify the amount of x, y, and z translation to perform to do the connection. In either case, the same window described above for zone coupling is presented with the word "Couple" replaced by the word "Connect". As with coupling, the user specifies a zone and boundary to connect to utilizing the connect data specified on the previous window.

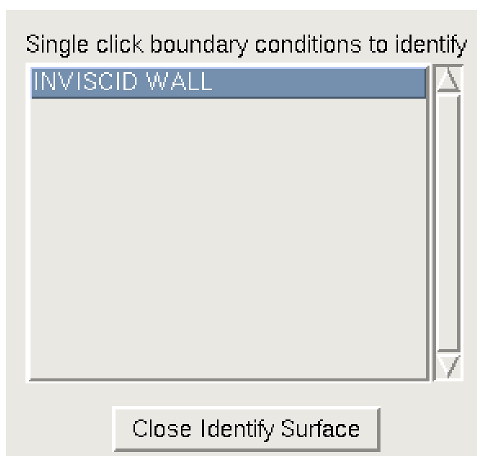
If the “Bleed Region” boundary condition is specified in the “Assign/Change Boundary Condition” window, the following window is displayed.



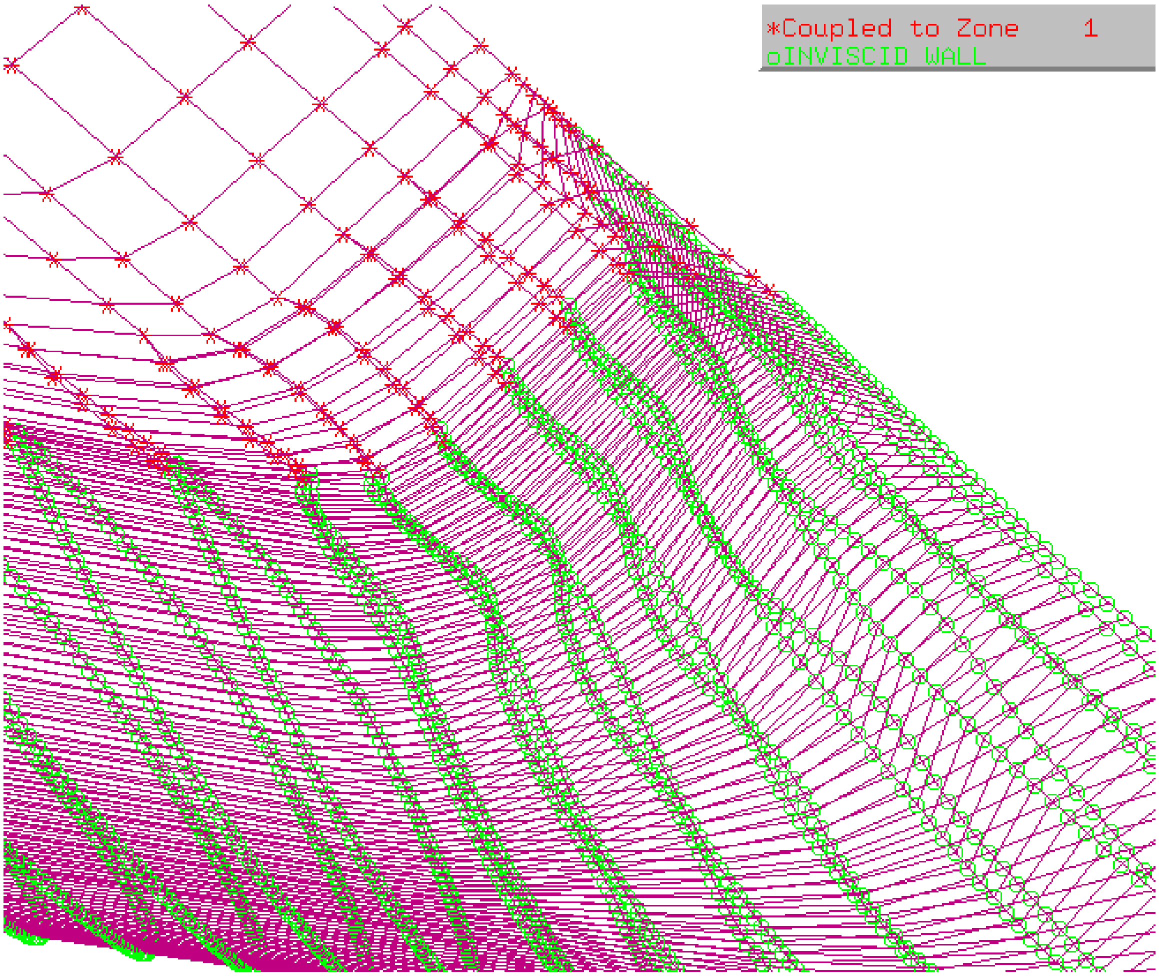
Up to 256 different bleed regions may be set in a given *.cgd* file. This window is used to simply enter an ID number for the Bleed Region that is being set on the currently-specified boundary condition region. Selecting the “Set” button will cause the defined region’s boundary condition to be changed to the bleed region ID entered here. Selecting the “Cancel” button ends the operation with no action being taken.

9.2 Identify Points

Selection of the “Identify Points ...” submenu initially causes a MADCAP prompt to be displayed asking the user to select from the graphics screen (see Section 2.6.3) a surface on which to graphically identify the points’ currently set boundary conditions. After responding to this select prompt, the following window is displayed to the user.



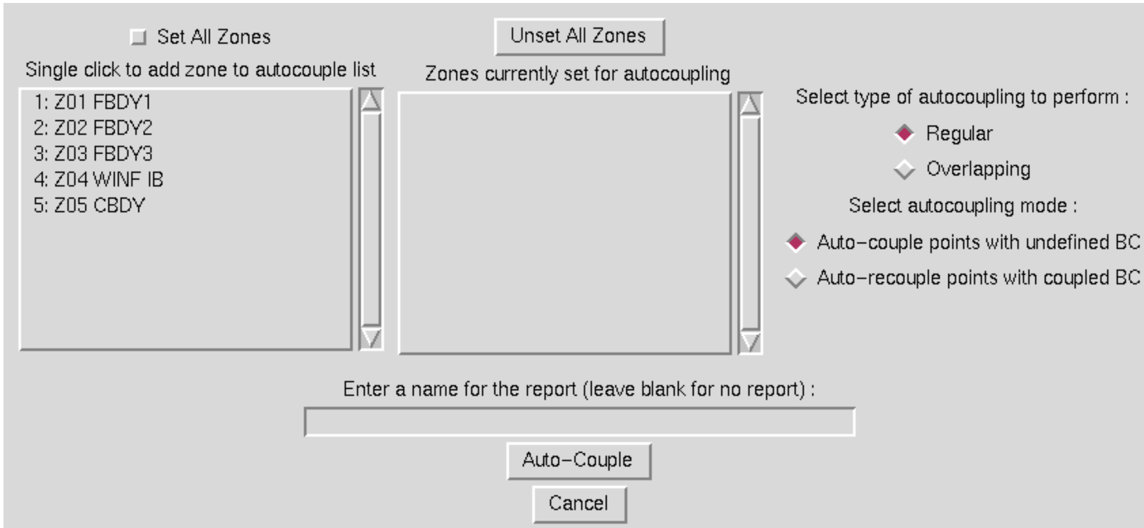
The list box contains the list of all boundary conditions existing on the selected surface. Single clicking on one of these boundary conditions will graphically identify the points on the selected surface, using markers in the graphics display, that correspond to that boundary condition. The Identify Points window will remain open, allowing points corresponding to other boundary conditions on the selected surface to be identified. When the “Close Identify Surface” button is selected, the window is closed and all identifying markers are erased from the display. A typical grid surface with identifying markers displayed is shown below.



Each boundary condition to be identified is given a unique color/marker combination from a set of three markers (*,o,x) and six colors (red, green, blue, cyan, magenta, yellow), resulting in 18 different possible unique markers before duplication. On the above figure, points tagged as inviscid wall are displayed as green circles while points coupled to zone 1 are displayed with red asterisks. A legend is provided in the upper right corner. Boundary condition identification markers are turned off while the image rotates or is being centered, but are turned back on when the operation is completed. This results in faster image manipulation when markers are displayed.

9.3 Auto-couple

Selection of the "Auto-couple ..." submenu causes the following window to be displayed.



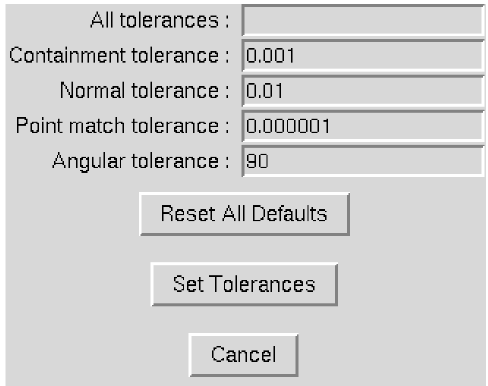
The list box on the left displays the full list of zones existing in the currently set boundary condition file. Any zone selected from this list is added to the list box on the right, which reflects the set of zones being set to perform automatic coupling in. Buttons are provided at the top of the window to either set all zones or clear the auto-coupling list. Both lists are vertically scrollable. The first set of radiobuttons to the right of the list boxes control whether the automatic coupling is performed in regular face-to-face coupling mode or overlapping mode. The second set of radiobuttons controls whether to auto-couple all undefined points, or to only re-couple the selected zones to whatever they were previously coupled to. An entry box is also provided in which a name for the report file generated by the autocoupling module can be entered. Leaving this box blank will cause no report to be generated. Once the set of zones to auto-couple has been set, the "Auto-Couple" button should be selected, and the window will disappear and MADCAP will proceed to autocouple the specified zones. To escape with no action being taken, the "Cancel" button should be selected.

The associated text command is:

```
bc automatic [couple | recouple] {mode regular | overlapping} \
   [zone {all | zonelist}] [report filename]
```

9.4 Set Tolerances

Selection of the "Set Tolerances ..." submenu causes the following window to be displayed.



The window consists primarily of five entry boxes in which to enter the desired tolerances to be used during coupling. There are four primary coupling tolerances — containment, normal, point match, and angular. All tolerances can be changed simultaneously by entering a value in the "All tolerances" entry, or individual tolerances can be changed with the separate entry boxes. The values shown above are the current default tolerances, which can be reset at any time by picking on the "Reset All Defaults" button. When the desired tolerances have been entered, the "Set Tolerances" button should be selected, and the new tolerances will be in effect for any future coupling. To escape with no action being taken, the "Cancel" button should be selected.

Containment tolerance controls how far outside the bounds of a cell a point can be and still be coupled to that cell. This is sometimes important along the edges of two non-point-matched boundaries. Normal tolerance controls how far a point must be projected normal to its own surface before finding an intersection with a cell in the face being coupled to. Angular tolerance controls how large the angle between the normals at the grid point being coupled and a candidate point to couple to can be. Point match tolerance is used in conjunction with point match coupling mode (Section 9.5), and specifies how far apart points can be and still allow points to be point-match coupled.

The associated text command is:

```
bc tolerance {containment | normal | point match | angular | all} \
   {default | tolerance}
```

9.5 Set Coupling Mode

Selection of the "Set Coupling Mode" submenu causes the following pull-down menu to be displayed.

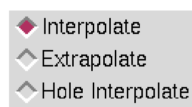
Four different modes can be set for performing coupling — bilinear fixed, bilinear best, bilinear full, and point match. Radiobuttons are provided for each of these modes. In Bilinear Best mode, coupling is attempted at each coupled or from type (for coupling from type all and undefined are the same) point on the boundary. If the point matched the from type, or if the point was coupled but the new coupling provides a better trifactors check than the old, the boundary condition is updated to the new coupling. In Bilinear Fixed mode, only from type points are coupled. Bilinear Full mode should be used only when the number of points to be coupled is small, due to the time- consuming nature of this mode. In this mode, a full coupling search is performed in place of the fast search methods normally used. This is rarely required, but may be useful to couple points that for some reason cannot be located using fast search methods. Since this method is time consuming and meant only for small detailed problem areas, it is not available in a “best” mode. Point match coupling is used to force every point to couple to its nearest counterpart in the coupled zone as if they were in fact the same point — that is, coupled exactly to a grid point and not to the interior of a grid cell. This is only useful when grids should in fact be point matched but are not exactly point matched due to small numerical differences.

The associated text command is:

```
bc couple mode {point match | bilinear | bilinear best | bilinear full}
```

9.6 Set Coupling Interpolation Mode

Selection of the “Set Coupling Interpolation Mode” submenu causes the following pull-down menu to be displayed.



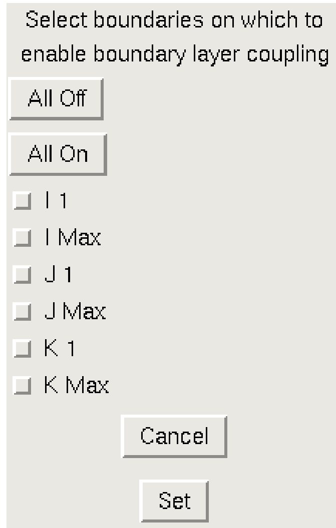
The Coupling Interpolation Mode controls how the coupling interpolation factors are set when a point couples to the exterior of a cell. When the mode is set to Interpolate, the interpolation factors are limited to the range 0. to 1., such that the point will couple to the edge of the target cell. If the mode is set to Extrapolate, the interpolation factors can be set less than 0. or greater than 1., so that during flow solver coupling, data may be extrapolated from the edges of the coupled cell. The Hole Interpolate mode is a special case for overlapping coupling. Normally, if a point is overlap coupled to the interior of a cell, the computed interpolation factors will be used even if some of the points on the cell are flagged as hole points. Setting Hole Interpolate mode will force the interpolation factors to project the coupling to the nearest cell face with no hole points, such that coupling to the interior of a cell containing hole points is prohibited.

The associated text command is:

```
bc couple mode {interpolate | extrapolate | hole_interpolate}
```

9.7 Set Boundary Layer Coupling

Selection of the “Set Boundary Layer Coupling” submenu causes the following window to be displayed.



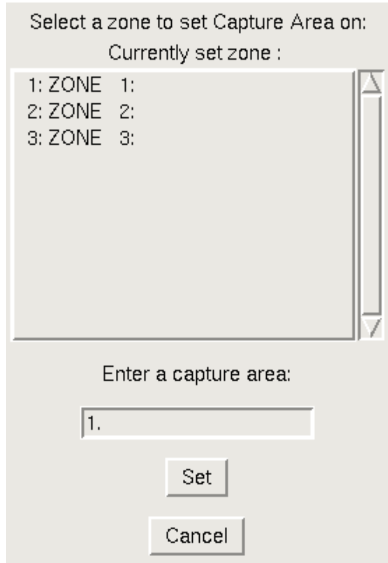
Boundary layer coupling should be used on edges of coupled faces that are walls, especially if the wall edge in the zone being coupled to is not point matched with the wall edge on the boundary being coupled. This ensures that the coupling parameters do not allow boundary layers to couple outside adjacent boundary layers or to be swallowed by adjacent zones by being entirely coupled to the first cell of the adjacent zone. For example, if the surface of a cylinder is described by the J1 surface in both zone 1 and zone 2, but zone 2 has a different set of circumferential points on the cylinder than zone 1, then boundary layer coupling should be enabled on the J1 boundary when coupling zone 1 IMAX to zone 2 I1, and vice versa. The window shown above simply provides checkboxes for each boundary, with buttons provided to turn on or off all boundaries. Boundary layer coupling will be performed on any edges where the checkbutton is set. Select the "OK" button to set the desired boundary layer coupling, or the "Cancel" button to escape with no action being taken.

The associated text command is:

```
bc couple mode boundary layer {on | off} {all | i1 | imax | j1 | jmax | \
   k1 | kmax}
```

9.8 Set Capture Area

Selection of the "Set Capture Area" submenu causes the following window to be displayed.



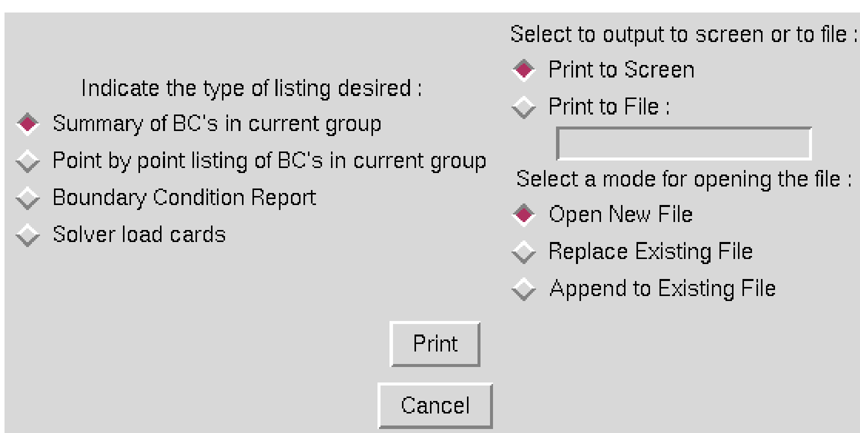
If a mass flow ratio is specified in the Wind-US input file for an outflow boundary condition, the *.cgd* file must contain a value for the inflow capture area for the associated zone. This window is used to enter a value for this capture area. The user should first select the zone to set the capture area in from the list of zones at the top of the window. Next, a value should be entered for the desired capture area. The value entered should be in the scale and units of the actual geometry being run. Selecting the "OK" button will cause this value to be stored in the specified zone of the currently set boundary condition file. To escape with no action being taken, select the "Cancel" button.

The text commands generated are:

`bc zone` *zone number*
`bc capture area [is]` *capture area*

9.9 Print BC Information

Selection of the "Print BC Information" submenu causes the following window to be displayed.



This window is used to set the type of boundary condition information to be reported on, and where to send the desired output. Three types of listing can be obtained — a summary of the boundary conditions in the current group, a point by point listing of the boundary conditions in the current group, or a Boundary Condition Report, which contains a description of the boundary conditions in the complete grid. The type of listing obtained is controlled by the radiobuttons on the left. On the right, the user may choose to print the listing either to the screen or to file. If the Print to File radiobutton is selected, the user must enter a file name for the listing and whether to open a new file, overwrite an existing file, or append to an existing file.

The associated text command is:

```
bc print {listing | report | summary}
```

If output is directed to a file, MADCAP precedes the `bc print` command with a file open command as follows:

`file open list` *filename* `[mode {new | replace | append}]`

9.10 Update Boundary Conditions

Selection of the "Update Boundary Conditions" submenu causes the boundary conditions set on the currently set boundary condition region to be written to the currently set boundary condition file. Until this action has been taken, the boundary conditions are only updated in program memory, and not to the actual *.cgd* file.

9.11 Cancel Boundary Condition Mods

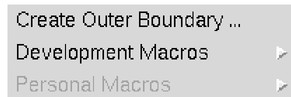
Selection of the "Cancel Boundary Condition Mods" submenu causes the boundary conditions that have been set on the currently set boundary condition region to be canceled out, and replaced with the original set of boundary conditions from the currently set boundary condition file. This action is non-recoverable, as all boundary condition information stored for the current region in program memory is restored to the file state existing before the set boundary condition operations were begun. To save the changes to file, use the "Update Boundary Conditions" submenu described in Section 9.10.

9.12 Release Boundary Condition File

Certain regions of program memory are allocated when a boundary condition file is first set, and used throughout the boundary condition setting process. This memory remains allocated and usable as long as a boundary condition file is set. If the MADCAP user has completed all desired boundary condition operations, it may be desirable to use the "Release Boundary Condition File" submenu to free up the memory associated with setting boundary conditions. This is never a required operation, but deallocates resources that may be required elsewhere. After selecting this submenu, no boundary condition file will be set, and the user must respecify a file before again setting boundary conditions.

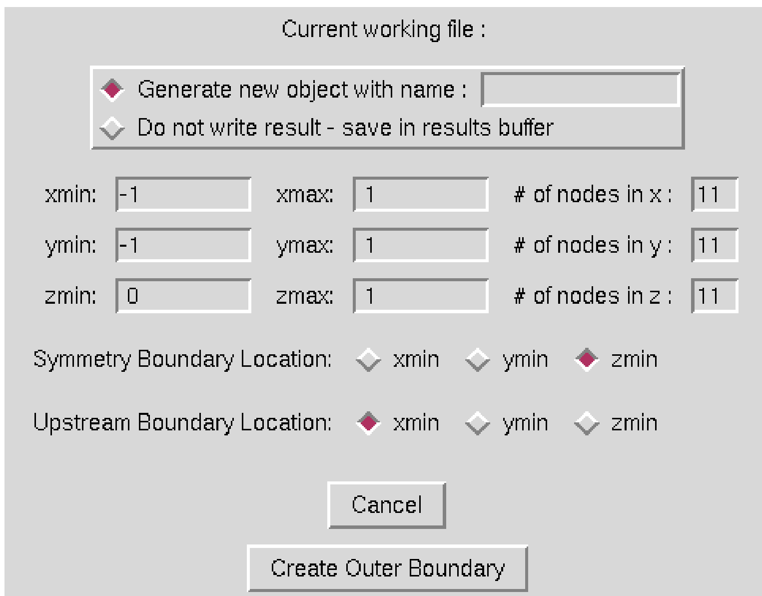
10 Macro Menu

The Macros menu provides access to scripts which automate certain grid generation tasks. Selection of the "Macros" menu item produces the drop down menu shown below.



10.1 Create Outer Boundary

Selection of the "Create Outer Boundary ..." menu item causes the following window to be displayed.



This macro is used to automatically define a box-shaped outer boundary for external flow cases. The dimensions of the box are entered in the field, and the location of the symmetry plane and upstream boundary surfaces is specified. The name of the newly created object can be entered at the top of the window.

10.2 Development Macros

Selecting the "Development Macros" menu item opens the following submenu.

The Development Macros menu provides access to a number of MADCAP scripts. These scripts or macros combine a number of MADCAP operations to create utilities that can greatly simplify the grid generation process. It is expected that the number of macros will continue to increase as new users assemble more macros. As these functions mature, they may be moved from the development macro menu into the regular MADCAP menus.

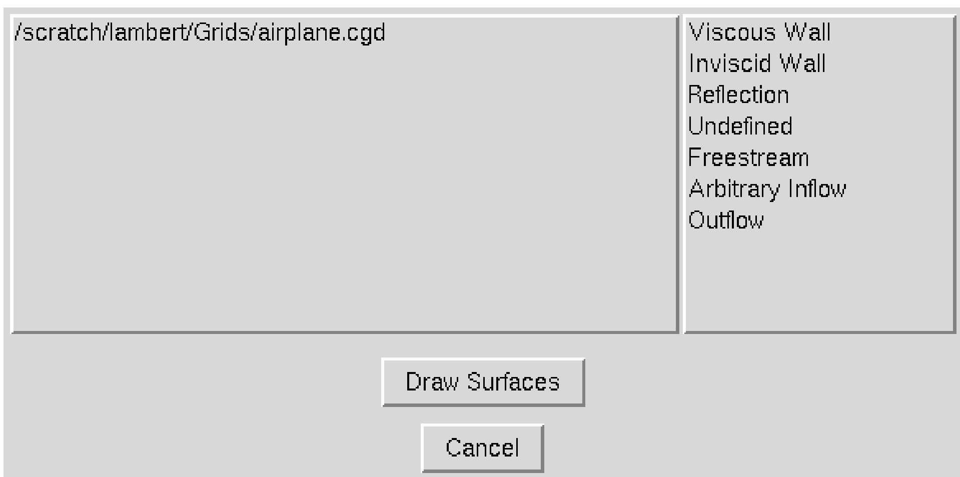
The development macros are currently under development, and may lack robustness in some cases or may only be applied under limited circumstances. They should be used with care. Even with these restrictions, they can be a powerful tool for the MADCAP user.

The available development macros are listed below.

- Draw CGD boundaries ...

10.2.1 Draw CGD Boundaries

Selecting the "Draw CGD Boundaries ..." item displays the following window.



This macro will display all the surfaces in all of the zones of a *.cgd* file by boundary condition. With this macro, all of the surfaces of a given boundary condition type (say viscous walls) in the entire *.cgd* file can be displayed in one easy operation. This is particularly useful for files with a large number of zones. This operation will work for structured, unstructured, or hybrid (mixed structured/unstructured) zone *.cgd* files.

10.3 Personal Macros

The "Personal Macros" menu allow the user to develop GUI interfaces to their own MADCAP macro scripts. The location of the users personal macros is specified through the environment variable `MADCAP_PERS_MACROS`. If this environment variable is set, MADCAP will search this directory path to look for any personal macros and display them in the menu list. For more information on creating your own MADCAP scripts, see the MADCAP reference guide.

Appendix A. MADCAP Keywords

The MADCAP text commands consist of primary keywords followed by optional keywords. The syntax of the commands follows the convention in use by the Wind-US and CFPOST keywords.

- File commands
  - file open {tmp | cgd | csf | list} *filename* [mode {new | append | \
    replace}]
  - file close {tmp | cgd | csf | list} *filename*
  - file axisdef *filename fsdir bldir wldir*
  - file convert {tmp} *from_filename* {csf} *to_filename*
- Draw commands
  - draw zone *zone_number*
  - draw surface {i | j | k} *index* [{*irange* | *jrange* | *krange*} \
    {*irange* | *jrange* | *krange*}] [color *index*]
  - draw surface *surfname* [zone *zonename*] [color *index*]
  - draw volume {bound | full} *irange jrange krange* [color *index*]
- View commands
  - view color {coloronbg | fgonbg}
  - view bgcolor *red green blue*
  - view fgcolor *red green blue*
  - view lightcolor *hue*
  - view axisorient {+x | -x | +y | -y | +z | -z} {+x | -x | +y | -y | \
    +z | -z}
  - view axismode {xyz | fsblwl}
  - view mode {asgenerated | wireframe | lighted | solid | hidden | edges}
  - view objmode {wireframe | lighted | solid | hidden | edges}
  - view blanking {grid | hole | off}
  - view reset
  - view displaylist {tmp | cgd | csf} *filename*
  - view sortmode {chron | alpha}
  - view lightsource {move | fix | save *filename* | restore *filename*}
  - view angle {+xaxis | -xaxis | +yaxis | -yaxis | +zaxis | -zaxis | \
    azimelev *angle1 angle2*}
  - view save {file *filename* | name *viewname*}
  - view restore {file *filename* | name *viewname*}
  - view delete name *viewname*
  - view clear
  - view sweep *time_interval* {IJK | IJ | IK | JK | I | J | K} \
    {{all | *start_index*} *sweep_increment* {forward | backward} \
    {cycle | reflect} ...}
  - view lines_per_page *nlines*
  - view messagebox
- Boundary Condition commands
  - bc file name *filename*
  - bc zone {*zone_number* | all | none}

- bc boundary {all | none | i1 | imax | j1 | jmax | k1 | kmax | olap | \
  Unnn}
- bc label {*label_number* | all | new | next | previous}
- bc couple mode {point match | bilinear | bilinear best | bilinear full}
- bc couple mode boundary layer {on | off} {all | i1 | imax | j1 | jmax | \
  k1 | kmax}
- bc couple mode {interpolate | extrapolate | hole_interpolate}
- bc couple mode connect {rotation angle *ax ay az* center *cx cy cz* \
  translation deltas *dx dy dz*}
- bc tolerance {containment | normal | point match | angular | all} \
  {*tolerance* | default}
- bc capture area [is] *area*
- bc automatic {couple | recouple} {mode regular | overlapping} \
  [zone {all | *zonelist*}] [report *filename*]
- bc subregion {I *range* J *range* K *range* | all}
- bc print {listing | report | summary}